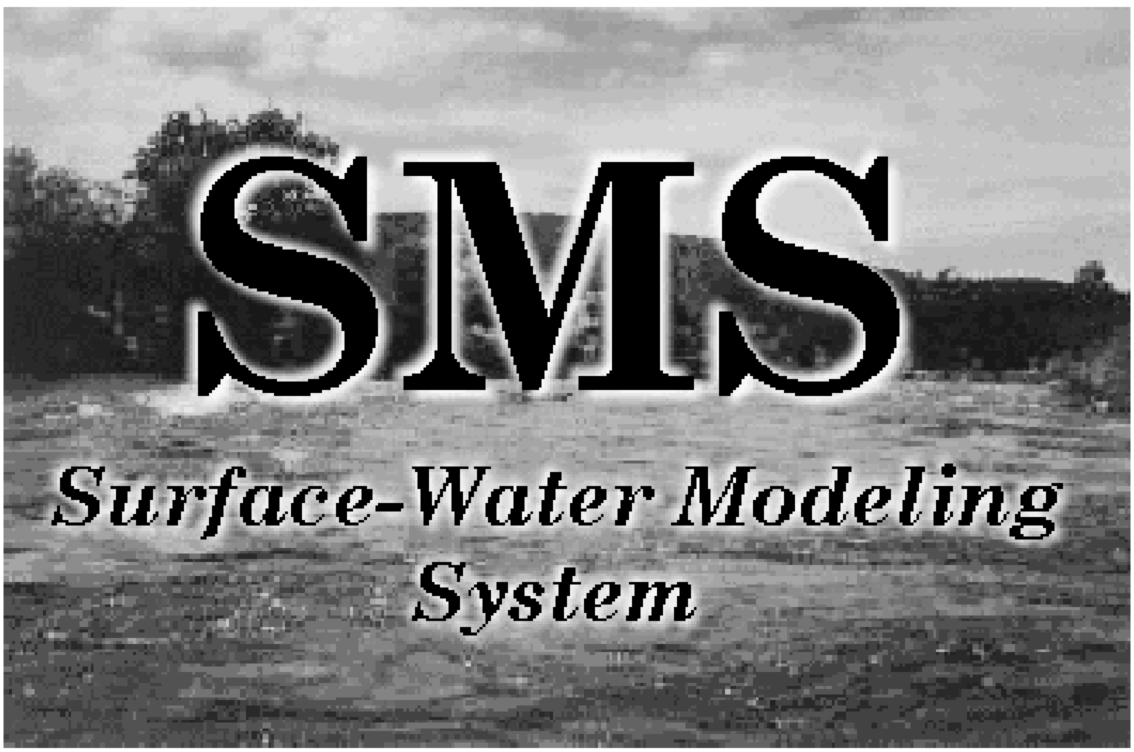


S M S T U T O R I A L

SMS 5.0 Tutorial

The software *SMS* is a product of the Engineering Computer Graphics Laboratory of Brigham Young University. For more information about this software and related products, call (801) 378-2812. FAX (801) 378-2478. Or write to:

Brigham Young University
Engineering Computer Graphics Laboratory
300 CB, Provo, Utah 84602

LESSON 1

Introduction

This document contains tutorials for the Surface-water Modeling System computer program (*SMS*). Each tutorial is meant to provide training on a specific component of *SMS*. It is strongly suggested that you complete the tutorials before using *SMS* on a routine basis.

SMS was developed by the Engineering Computer Graphics Laboratory at Brigham Young University in cooperation with the U.S. Army Corps of Engineers Waterways Experiment Station (WES) and the Federal Highways Administration (FHWA). It was designed to be used in conjunction with the *TABS-MD* computer programs maintained by WES, and/or with the *FESWMS-2DH* and *WSPRO* computer programs sponsored by the FHWA, which use the finite element method to solve complex systems of equations. *SMS* is a pre- and post-processor for two-dimensional shallow open-water models such as rivers, bays, and estuaries. This means that a finite element mesh can be created within *SMS*, and the solutions from *TABS-MD* and *FESWMS-2DH* can be viewed in *SMS*, but the actual analysis is performed by the finite element analysis programs. *SMS* is well suited for the construction of large, complex meshes of several thousand elements.

Please note that in this tutorial, reference to a menu item will be as follows: *Menu* | *Menu-Item*. For example: *File* | *Quit* indicates to select the *Quit* item from the *File* menu

1.1 REFERENCE MANUAL

Accompanying this tutorial is the *SMS Reference Manual*, which describes the *SMS* interface. It is suggested that the tutorials be read prior to reading through the reference manual. However, there are some parts of the tutorial which refer to the reference manual. In these cases, the particular section of the reference manual should be read if a better understanding is needed.

1.2 SUGGESTED ORDER OF COMPLETION

Most of these lessons are developed for two dimensional surface water modeling. If you want to use the SMS for its River Module and WSPRO interface, you may want to examine the first tutorial (Lesson 2), and then go on to Lesson 11. For other uses of SMS, we recommend that you start with the first tutorial (Lesson 2). It is entitled Overview of SMS, and gives a quick tour of the general features of *SMS*. It also describes the basic tools available in *SMS* for generating a mesh from data points. Lesson 3 on Mesh Editing should be completed next. From there it is suggested that Lesson 4 on Basic RMA2 Analysis or Lesson 7 on Basic FESWMS Analysis be completed. Lesson 10, which concentrates on using the post-processing capabilities to view the solutions uses the *RMA2* results from Lesson 5, but the techniques presented are almost identical for post-processing *FESWMS* solutions. Lesson 10 should be reviewed after covering the appropriate pre-processing and analysis section. All required files are provided, so these tutorials can be completed using either a demonstration or licensed copy of *SMS*.

1.3 DEMO vs. WORKING MODES

The interface for *SMS* is divided into three separate modules. A module is provided for each of the basic data types supported by *SMS*. As the user switches from one module to another module, the *Toolbox* and the menus change. The modules allow the user to focus only on that portion of the interface which is necessary to solve a particular problem. Some of the modules contain interfaces to a specific model such as *RMA2*. Such interfaces are typically contained within a single menu.

Since some users may not require all of the modules or model interfaces provided in *SMS*, modules and model interfaces can be licensed individually. In a demo version the icons for the unlicensed modules and the menus for unlicensed model interfaces are dimmed and cannot be accessed.

Although you may have licensed only a portion of the *SMS* interface, a complete set of mesh tutorials are included in this document. This will allow you to test and evaluate the model interfaces you have not licensed, in case you are interested in licensing them at some future date. When you start *SMS*, only the licensed modules will be enabled. To enable unlicensed modules for use with the tutorials, the user should enter demonstration mode. To go into demonstration mode, do the following:

1. Select *File | Demo mode.*

2. Select *OK* to the prompt, *Are you sure you want to delete everything?*

Once in demonstration mode, the *Save* and *Print* commands in all menus have been disabled. In demonstration mode, all functions of all modules and interfaces are available, with the exception of saving, exporting, and printing.

Once you are finished with the tutorial, you can go back to normal mode by doing the following:

1. Select *File | Normal mode*.

2. Select *OK* to the prompt, *Are you sure you want to delete everything?*

1.4 TABS-MD vs. FESWMS-2DH

The *TABS-MD* code was originally developed by Norton, et al. (1973) of Resource Management Associates, Inc. of Davis, California. The *TABS-MD* suite, which consists of several separate programs, can be used to calculate water surface elevations, water depths, flow velocities (*RMA2*), contaminant migrations (*RMA4*), and sediment transport (*SED2D-WES*) of open flow systems. It can be used for either steady state models (flow at a single time) or dynamic models (flow that varies with time).

The *FESWMS-2DH* code was developed for the Federal Highway Administration by David C. Froehlich. As with the *TABS-MD* programs, the *FESWMS-2DH* programs can be used to calculate steady state or dynamic water surface elevations and flow velocities. However, *FESWMS-2DH* differs from *TABS-MD* in that it easily handles flow control structures, such as culverts and weirs, and allows easy integration of flow obstructions such as bridge piers, directly in the two dimensional mesh. The *TABS-MD* programs require the use of one dimensional elements to approximate flow control structures and mesh modification to represent piers.

Both models support wetting/drying procedures for regions of the mesh. *TABS* provides the additional option of modeling marsh porosity in the mesh.

The finite element meshes and associated boundary conditions necessary for *TABS-MD* and *FESWMS-2DH* modeling may be created within *SMS*, then saved to model specific files containing the geometric description of the model and controlling boundary conditions. These files are then used to perform the analyses. Resulting solution files can be read into *SMS* to generate vector plots, color-shaded contour plots, time-history diagrams, and solution animation sequences. *SMS* can also be used with other finite element computer programs, provided they read and write files in either the *SMS*, *TABS-MD*, or *FESWMS-2DH* format.

LESSON 2

Overview of SMS

2.1 INTRODUCTION

This tutorial provides a general overview of creating a mesh in *SMS* and should be completed prior to beginning any of the other tutorials. It describes each of the major components of the interface (menus, *Toolbox*, windows, etc.) and gives a brief introduction to the *SMS* modules.

2.2 GETTING STARTED

Before beginning this tutorial you should have installed *SMS* on your computer according to the directions in the *SMS Installation Guide*. If you have not yet installed *SMS*, please do so before continuing.

These tutorials require a complete set of model interfaces in *SMS* (all interfaces are included). If you have purchased a version of *SMS* which does not contain all of the modules or model interfaces, or if your are evaluating the software, you should use the demonstration mode to complete this tutorial (see the *DEMO vs. WORKING MODES* section in Lesson 1 for more information).

To start up *SMS*, follow the instruction for the appropriate machine:

2.2.1 PC

For WIN 3.1/ 3.11/ NT 3.x start *SMS* by double clicking on the *SMS* icon from within the *Windows Program Manager*.

For WIN 95/ NT 4.0 start *SMS* by opening the *Start* menu, scroll to *Programs* and then scroll to *SMS* and click on it.

2.2.2 UNIX

Go to the *SMS* directory and type **sms** at the command line.

2.3 THE SMS SCREEN

Once you start the program you should see a menu bar and three windows as shown in Figure 2-1. The layout of the items in the windows may differ slightly from what is shown, depending on whether you are running the PC version or the UNIX version of *SMS*. The *SMS* screen is divided into four main sections: the *Graphics Window*, the *Toolbox*, the *Edit Window*, and the *Menu Bar*. An additional window used for two dimensional plots, the *Plot Window*, appears when required.

There are four sections of tools in the *Toolbox*. In the first section, three icons at the top represent the different *SMS* modules. The second section is a set of tools used for manipulating the display. The third section is a set of module-specific tools. These allow selection, creation and editing of entities specified to the current module. The fourth section contains macros, which are shortcuts for frequently used menu commands.

The *Edit Window* is used to edit coordinates, display prompts and print help messages.

- Using the mouse cursor, click on the icons at the top of the *Toolbox* to switch between modules.

As you change modules, notice that some of the menus and some of the tools in the *Toolbox* change. To keep the interface as simple as possible, you are only presented with the tools and menus associated with the current module.

- Move the mouse cursor to one of the menus, press the mouse button, and keep it depressed as you move the cursor down the menu highlighting the commands.

Note that a description of each command is presented in the right side of the *Edit Window* at the bottom of the screen. This is part of the on-line help provided in *SMS*. Help messages are also displayed as you move the cursor over items within a dialog box.

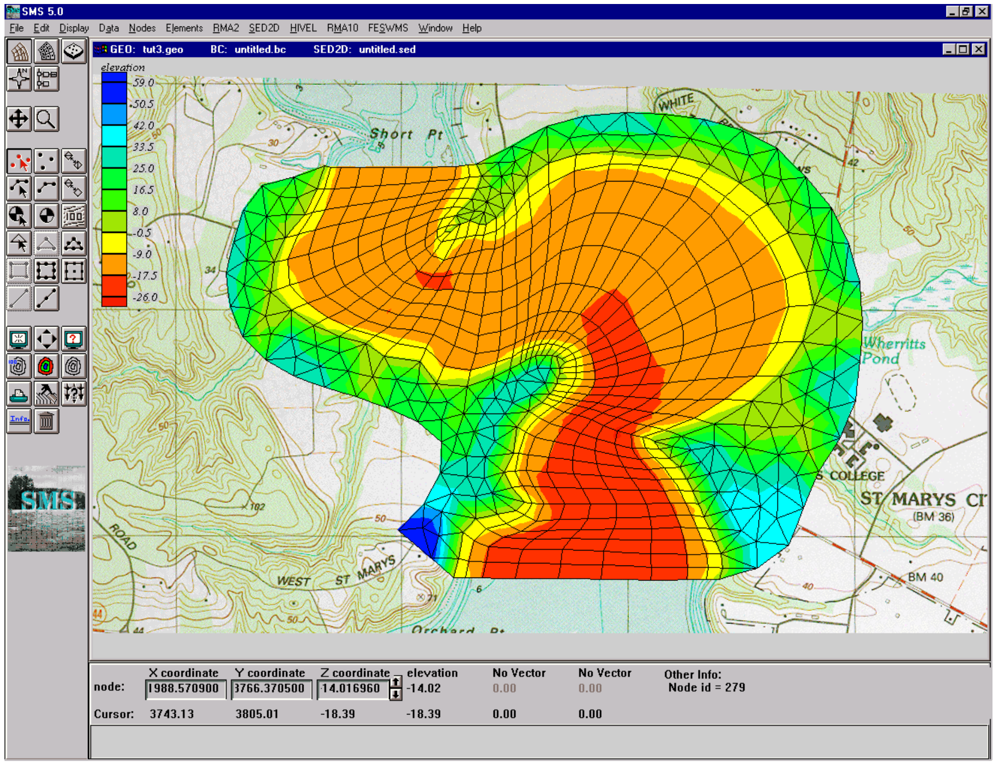


Figure 2-1. The SMS *Screen.*

The first step in creating a mesh is defining the boundaries. *SMS* creates interior elements within these bounds according to user defined parameters. Scatter point data can then be used to interpolate bathymetric information (elevations) into the newly formed mesh.

2.4 Importing the Background Image

To define the mesh boundaries, import a digital image of the site being modeled. For this example the image was created by scanning a portion of a USGS quadrangle map on a desktop scanner. The image was saved from the scanning software to a TIFF file. Once the image is imported to *SMS*, it can be displayed in the background as a guide for on screen digitizing and placement of mesh features.

2.4.1 Reading the Image

To import a TIFF image:

1. Select *File* | *Import*.

2. In the *Select Import Format* dialog select the TIF/GIF option and click on the *OK* button or press the ENTER key. The *Read File* dialog will open.

3. In the *Read File* dialog, locate and open the directory entitled s*ms/tutorials*.

4. Find and highlight the file “tut1.tif”, and click the *Open* button.

5. This will open the register image dialog.

When a TIFF file is first imported it must be registered. Registering a TIFF file involves selecting three points on the image and entering the real world coordinates of those points. These points are used to stretch and position the image when it is drawn on the screen so that it is drawn in the correct location.

Register point values may be entered in manually or a world file can be imported that already has the set values for the register points. We will import our register points from a world file.

6. In the register dialog click on the *Import World File* button. This opens the *Read File* dialog.

7. In the *Read File* dialog, locate the file “tut1.tfw”, highlight this file and then click on the *Open* button.

   This will input values for the three register points and update the positions of the register points (red cross-hairs).

8. Click the *OK* button or press the ENTER key.

You should see a progress bar at the bottom of the edit window indicating that the image is being resampled. Now that the image is imported, it will appear each time the screen is refreshed. All other objects are drawn on top of the image.

Often it is desired that the image be manipulated and resampled to enhance the picture.

1. Select the *Zoom* tool from the second section of the *Toolbox*.

2. Drag a box around the main portion of the river. It should look like Figure 2-3.

The image is quite blurry and many details are hard to see.

3. Select the *Map Module* icon from the module section of the *Toolbox.*

4. Select *Images* | *Resample*. This will resample the tif image to the viewing window and enhance the resolution.

2.5 Feature Objects

We are now ready to begin constructing the conceptual model. Conceptual models are constructed using *feature objects* in the *Map* module. Feature objects in *SMS* include points, nodes, arcs, and polygons.

Points are xy locations that are not attached to an arc. Points have unique ids and can be assigned attributes. Points are typically used to force the creation of a mesh node at a specific location, or to specify mesh density in a specific region.

Arcs are sequences of line segments or edges which are grouped together as a single "polyline" entity. Arcs have unique ids and can be assigned attributes. Arcs can be grouped together to form polygons or used independently to represent linear features such as channel edges. The two end points of an arc are called "nodes" and the intermediate points are called "vertices". Nodes have unique ids and can be assigned attributes. Vertices are used solely to define the geometry of the arc.

Polygons are groups of connected arcs that form a closed loop. A polygon can consist of a single arc or multiple arcs. If two polygons are adjacent, the arc(s) forming the boundary between the polygons is shared (not duplicated).

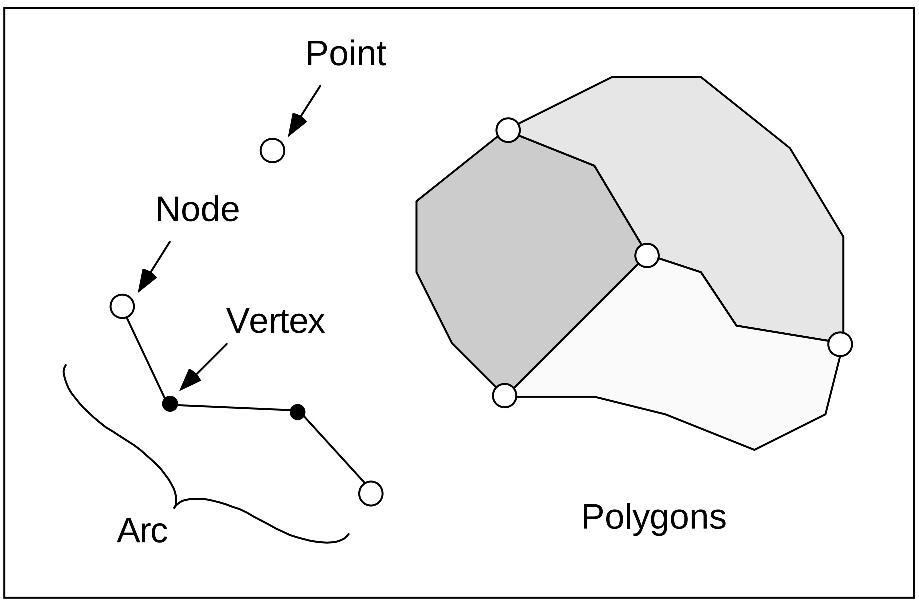


Figure 2-2 Feature Objects

Feature objects can be grouped into sets called *coverages*. Each coverage represents a particular set of data. Our conceptual model will consist of one coverage: a coverage defining the geometry of the river regions and the flood bank. As we go along in this tutorial you will load new coverages over the existing coverage. The new coverage will become active and the old coverage will be dimmed and inactive.

2.6 CREATE FEATURE ARCS

A set of feature arcs and points can be created to show topographically important features such as channel bottoms, ridges, material region boundaries, flow control structures, etc. This can be done by converting data from a DXF file, on screen digitizing using a background mesh, or on-screen digitizing from a georeferenced TIFF image. We are using this last method.

1. Select the *Create Feature Arcs*  tool from the *Toolbox*.

2. Using the mouse click on various points along the left edge of the river, double click the mouse button when you reach the end. (see Figure 2-3)

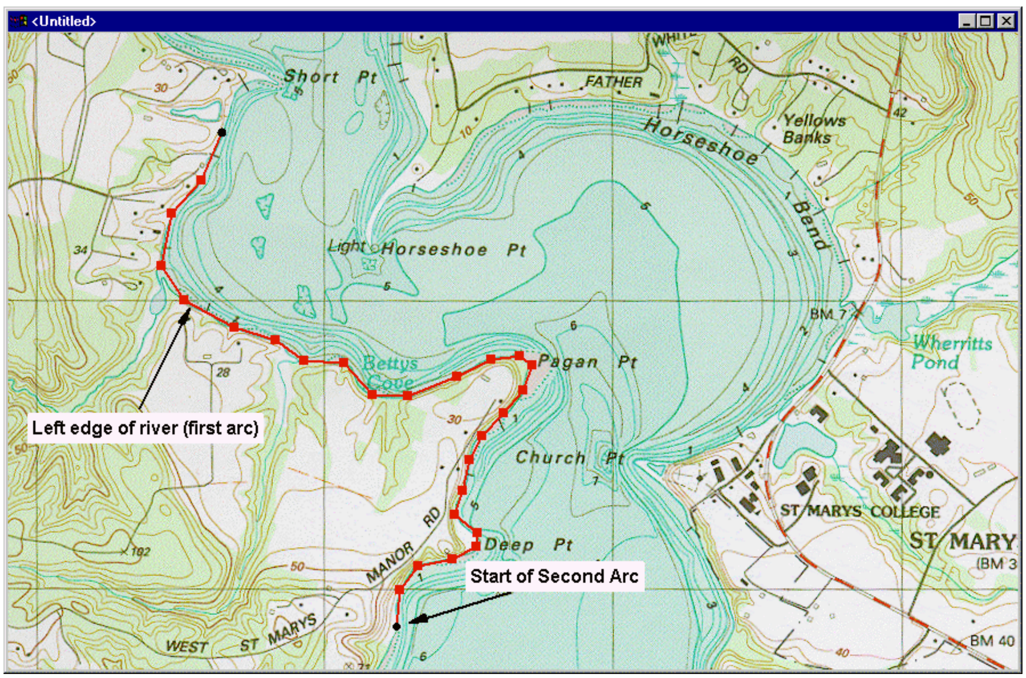


Figure 2-3 Creation of Feature Arcs

Note: As you are creating vertices, if you make a mistake and wish to back up, press the BACKSPACE key. If you wish to abort the arc and start over, press the ESC key.

1. Create Arc #2 across the river. Start by clicking on the last node from the first arc and continue to the other side of the river. (see Figure2-4)

By clicking on a node of an existing arc, the new arc will share the end point.

2. Make two more arcs, one going up the right bank and the other across the top of the river. Your screen should now look just like Figure2-4.

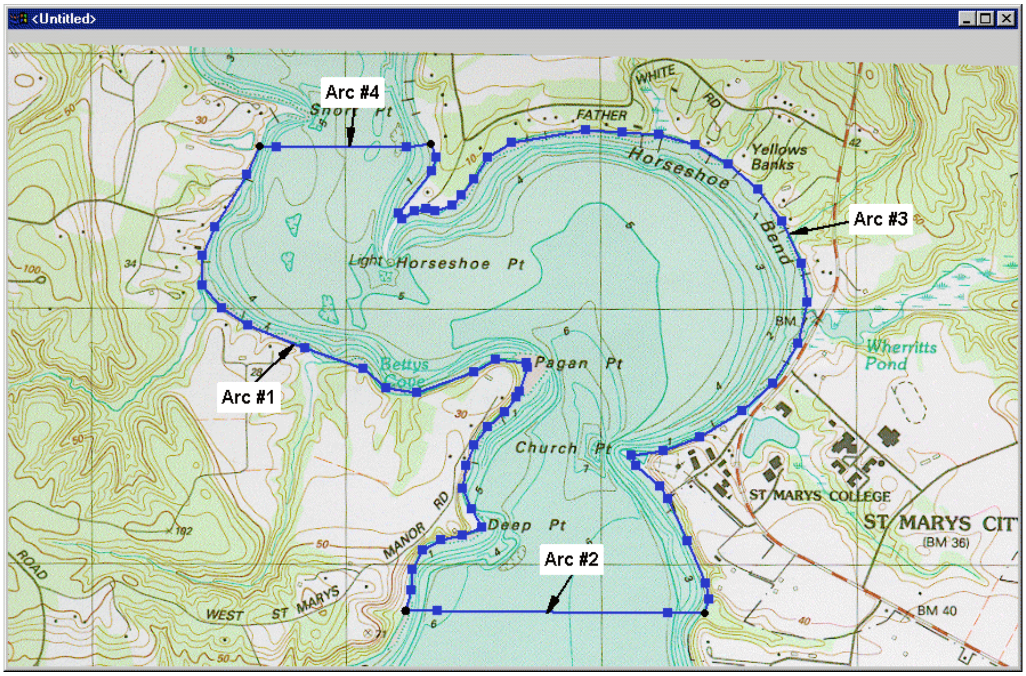


Figure2-4 Feature Arcs

Normally you would continue to create arcs until all features, material zones and meshing regions are defined. To save time this has been done for you.

1. To read this data, select *File* | *Open* and select the file "tut1_a.map". This map file will be placed as a new coverage over your existing feature arcs.

2.7 MANIPULATING COVERAGES

As stated at the beginning of this tutorial, feature objects can be grouped into coverages. When a new set of feature objects is loaded, it is placed in a new coverage. This coverage is made active, and the old set of feature objects are dimmed. Coverages can be manipulated through the coverages dialog.

1. Select *Feature Objects* | *Coverages*. A list of all coverages appears and each coverage is assigned a name and may be made visible or active.

2. The coverage that is indicated as active (arcs defined tut1_a.map) is the new coverage. Click the *hide all* button and then make just the active coverage visible by selecting it and selecting the visible check box. You could also just delete the unwanted coverages by selecting them and clicking on the *Delete* button.

3. Click *OK* to accept the changes made in the *Coverages* dialog or click Cancel to exit the dialog without saving the changes.

As the tutorial progresses you might want to hide or delete old coverages. In this tutorial, coverage names correspond with the filenames.

2.8 REDISTRIBUTE VERTICES

To create the feature arcs, you simply clicked on certain points of interest on the image. The number and distribution of vertices along an arc is not necessarily uniform. Element density in a mesh created from feature objects matches the density of vertices along the feature arcs. To evenly distribute the density of elements in the generated mesh, the vertices can be redistributed along an arc. Vertices can be added one at a time using the *Create Vertex* tool or entire arcs can be refined with a single command.

Refer to Figure 2-5 to perform the following vertex redistribution operations.

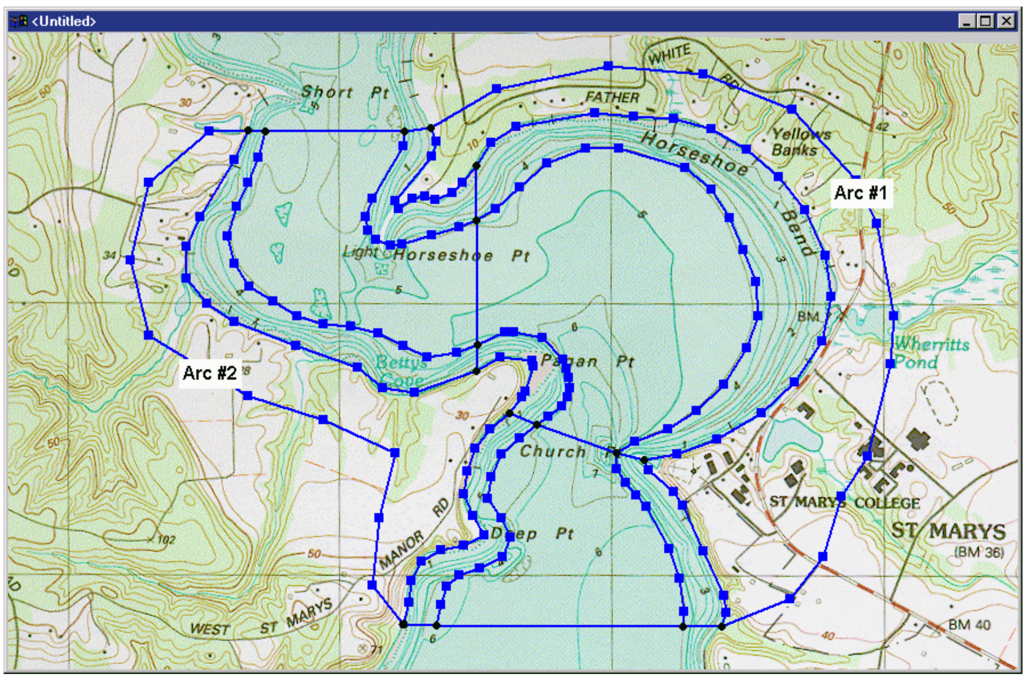


Figure 2-5 Redistribution of Vertices along arcs

1. Select the *Select Feature Arcs* tool from the *Toolbox*.

2. Click on the arc to the far right (arc #1). Then choose *Feature Objects* | *Redistribute Vertices*.

   In the *Redistribute Vertices* dialog, you are given information about the number of arc segments, average spacing and other items.

3. Select the *specified spacing* option and enter a value of 470. Click *OK* and the vertices will be redistributed along the arc.

4. Do the exact same thing for the arc to the far left (arc #2).

This would normally be done for each arc. Arcs on opposing sides of a patch will create a better patch if they have equal number of vertices. For these arcs, vertices can be redistributed here or in the patch dialog that will be explained later in this tutorial. If defined at this point then it is better to give a specified number of arc segments rather than a spacing interval.

Multiple arcs can be selected at one time by holding down the SHIFT key while selecting the arcs. In the *Redistribute Vertices* dialog, a combination of all the arcs data will be displayed, and the parameters chosen will affect each arc individually.

Open the file "tut1_b.map". This file has all the arcs distributed and is ready for further development.

2.9 DEFINE POLYGONS

Using the previously defined arcs you will create polygons bounding the material zones. Polygons can be made by selecting the arcs bounding a specific polygon or if no arcs are selected then polygons will be made from all the existing feature arcs.

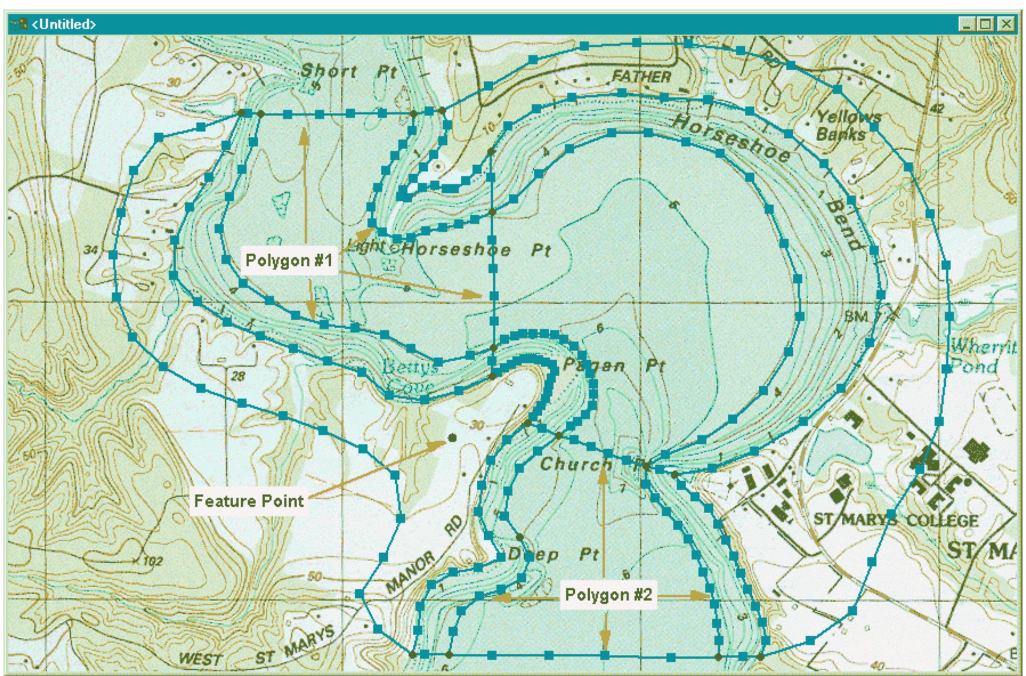


Figure 2-6 Defining polygons & Meshing

1. Select the *Select Feature Arcs* tool from the *Toolbox*. Holding down the SHIFT key, click on each of the four arcs that will make the polygon #1 in Figure 2-6.
2. Select *Feature Objects | Build Polygons*. Those four arcs now bound a polygon.
3. This can be seen by selecting the *Select Feature Polygon* tool from the *Toolbox* and clicking in the polygon.

Rather than building all the polygons individually, all polygons can be built at once.

1. Make sure no arcs are selected
2. Select *Feature Objects | Build Polygons* and select *OK* to use all feature arcs.

All the material zones are now delineated by polygons. Some of the material zones are further broken into divisions to aid in creating a better quality mesh.

2.10 ASSIGN MESHING PARAMETERS

With polygons, arcs and points created, meshing parameters can be assigned. *SMS* uses two meshing techniques, patch and adaptive tesselation. Adaptive tesselation is the default technique and will work for all polygon shapes. Patching tries to fill the polygon with elements oriented to the patch sides. To create a patch, a polygon must have 3 or 4 sides. Each side is defined by 1 or more arcs (see Figure 2-7).

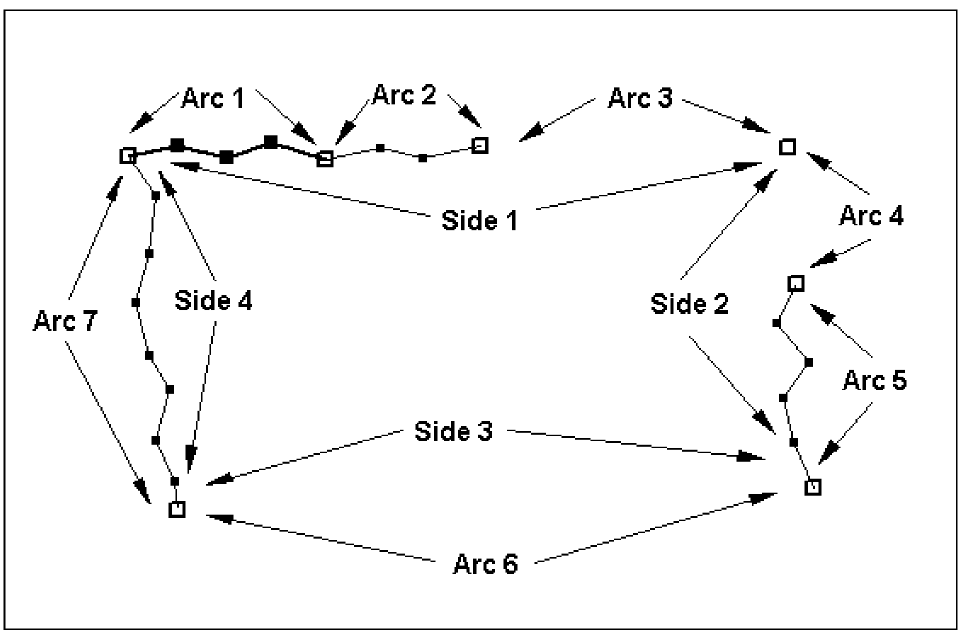


Figure 2-7 Requirements for a patch

With feature points SMS provides the option of specifying the element size for those elements that will be created around the point.

1. Select the *Select Feature Point Node*  tool from the *Toolbox* and double click on the labeled feature point on the left bank (see Figure 2-6).

2. This brings up the *Feature Point/Node Attributes* dialog. Select the *Refine Point* button , enter a value of 300 and select *OK* or hit the ENTER key. Now when the mesh is created all elements connected to this point will have an edge length of 300.

Now you are ready to assign meshing techniques to each polygon. Remember that only polygons with 3-4 sides can be patched.

1. Select the *Select Feature Polygon*  tool and double click on the polygon located at the bottom of the river in the center (labeled polygon #2 in Figure 2-6)

2. This brings up the *Feature Polygon Attributes* dialog.

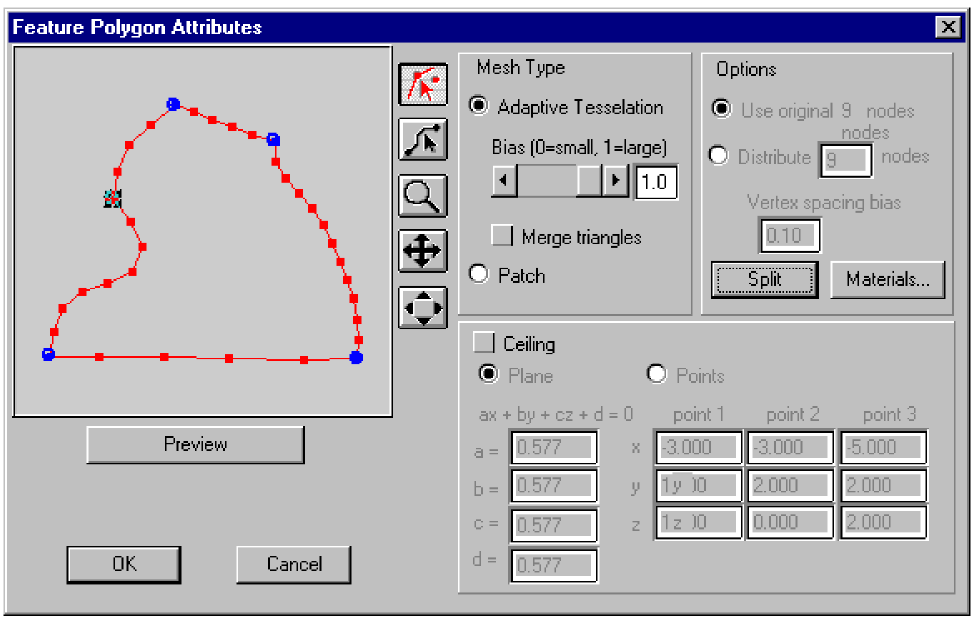


Figure 2-8 Feature Polygon Attributes Dialog

3. Select the *Select Feature Point Node*  tool from the attributes dialog.

4. Click on the node at the center of the left arc as seen in Figure 2-8. Then select the *Merge* button. This will merge the two arcs together and treat them as one side to the patch. Notice that the *Patch* option is now undimmed.

5. Select the *Patch* option.

6. Hit the *preview* button and you will notice that the elements in the mesh are a little crooked.

7. Select the *Select Feature Arc*  tool from the attributes dialog.

8. Select the lower left arc, and select the option to Distribute 8 nodes located in the right side of the dialog.

9. Assign a bias of *1.2*. Hitting preview again will show the updated mesh.

10. Press the ENTER key or select the *OK* button.

    -If you select adaptive tesselation then you can adjust the size bias to make smaller or larger elements.

Open the file "tut1_c.map". This has all the meshing parameters defined.

You will notice that throughout this tutorial, drawing objects (labels and arrows) have been provided. To delete these objects:

1. Select the *Select Drawing Objects*  tool from the *Toolbox*.

2. Select the drawing object that you want deleted and press the DELETE key or one of the other deletion methods. You can select multiple drawing objects by holding the SHIFT key as you click them.

2.11 APPLYING BOUNDARY CONDITIONS

Boundary conditions can be assigned to feature arcs, points and polygons. Arcs may be assigned flow, head, or flux status. Points may be assigned velocities or head values and polygons may be assigned ceiling elevation functions.

1. Select the *Select Feature Arcs*  tool and double click on the arc at the top of the map that crosses the river.

2. In the *Feature Arc Attributes* dialog select the *Specified Flow* button and assign a flow of 5000 cfs. Select the *OK* button or press the ENTER key.

3. Repeat the last two steps for the other two arcs along the top of the section (those arcs across the left and right channels). Assign 1000 cfs to these arcs.

4. Select the *Select Feature Arcs*  tool and double click on the arc at the bottom of the map that crosses the river.

5. In the *Feature Arc Attributes* dialog select the *Specified Head* button and assign a value of 6 ft.

6. Repeat this for all the arcs at the bottom of the section. Assign values of 6 ft for the left and right channel.

The boundary conditions are now defined.

2.12 DEFINE AND ASSIGN MATERIAL TYPES TO POLYGONS

Polygons may also be assigned specific material types. Elements created to fill the polygon will be assigned the polygon's material type, and assume properties assigned to that material.

1. Select the *Select Feature Polygon*  tool and double click on the far right polygon (flood plain area). This will open the *Feature Polygon Attributes* dialog.

2. Choose the materials button which opens the *Materials Editor* dialog. This allows you to assign materials to the chosen polygon. Select the material "flood plain" and select the *Close* button or press the ENTER key. Click on *OK* or press the ENTER key to exit the *Feature Polygon Attributes* dialog.

3. Assign material properties shown on Figure 2-9 to all the polygons. Holding the SHIFT key, select all the polygons with similar material properties, then select *Feature Objects | Attributes* or double click on a polygon after the last polygon has been selected. In the *Feature Polygon Attributes* dialog select the materials button and assign the material type.

4. Select the *Display Options*  macro. This opens the *Feature Objects Display* dialog.

5. Turn the polygon toggle on and select to show material types.

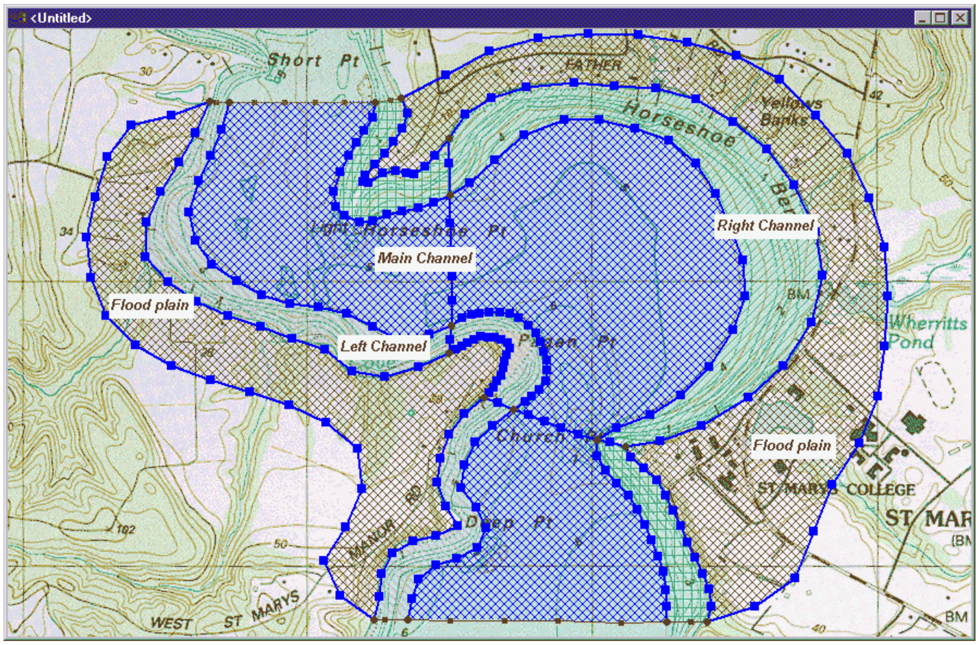


Figure 2-9 Polygons with defined material types.

2.13 CONVERT FEATURE OBJECTS TO MESH

With all the boundary conditions assigned, and meshing techniques chosen we are ready to create elements from the polygons.

1. Select *Feature Objects* | *Map ->2D Mesh*.

2. Choose the quadratic elements option.

3. Click on the *OK* button or press the ENTER key.

The mesh will now be generated according to the preset conditions. With the mesh created it is often desirable to delete the feature arcs or hide them. To hide the feature arcs:

1. Select the *Display Options*  macro from the *Toolbox*.

2. Turn all of the feature object toggles off and then click on *OK* or press the ENTER key.

With the feature objects hidden the mesh can be manipulated without interference, but feature objects are still available if mesh reconstruction is desired.

2.14 Editing the Feature Object Mesh

When the feature object mesh is produced it may not be exactly as you want it. To refine the mesh you can use many of the editing techniques outlined in the following lesson, or you can change the meshing parameters in the conceptual model and regenerate the mesh.

2.15 INTERPOLATING INTO THE MESH

A mesh is now created that bounds the zones outlined by the feature objects. The generated mesh has no topographic information and this must be assigned to it. This will be done by interpolating a set of scatter point bathymetric data onto the mesh.

1. Select the *Scatter Point Module*  icon from the *Toolbox*.

2. Select *File* | *Open* and choose the file "stmary.sup". This file contains the scatter point geometry and bathymetry.

3. Select *Interpolation* | *Interp. Opts.*.

SMS has many different interpolation techniques. For a complete explanation of each method see the *SMS* reference manual.

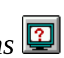
4. Select *linear interpolation* and press the ENTER key or click the *OK* button.

5. Select *Interpolation* | *To Mesh*.

6. Click on the *Map Elevations* option and click *OK* or press the ENTER key.

The scattered data is then triangulated and values interpolated for each node in the existing mesh. The status bar in the edit window will show how the interpolation is proceeding. Interpolation to the mesh should take 15-60 seconds.

Like the feature objects, after the scattered data is used, it is nice to hide or delete it. To hide scatter point data:

1. Select the *Display Options*  macro from the *Toolbox*.

2. Turn off the scatter point toggle and then click the *OK* button or press the ENTER key.

With the scatter point data hidden the mesh can be manipulated without interference, but scatter point data is still available if re-interpolation is desired.

1. Select the *Mesh Module*  icon from the *Toolbox*.

2.16 Saving all Data as a Superfile

When a mesh is created there are many sets of data that can be saved. There is the two dimensional geometry, the material settings, bathymetric data and image files. These can be saved individually or they can be saved together as one superfile for ease in saving and opening of mesh data.

1. Select *File* | *Save*.

Notice that all of the data sets contained in the mesh are selected. You can choose which sets to be saved and whether a super file is to be created by clicking on the toggle boxes next to each data type. We will save a superfile containing all of the data types.

1. In the *Prefix for all files* window enter the name tut1.

2. Click the *Update* button, then Click *Save* or press the ENTER key.

2.17 Saving the Geometry for RMA2 and FESWMS analysis

Once the mesh has been created, the geometry data should be saved to a file. *SMS* can output the geometry into various file formats. See the *SMS* Reference Manual for a description of various file formats. To save your mesh to a GFGEN geometry file for use with *RMA2*:

1. Select *RMA2* | *Save Geometry*.

2. Specify the geo file name. The file "tut1.geo" has been supplied, but may be overwritten.

3. Click the Save button or press the ENTER key.

Notice that the *Graphics Window* title bar is updated with the name of the geometry file. To save the same mesh to a *FESWMS* geometry file:

1. Select *FESWMS* | *Save Simulation*.

2. Enter tut1 in the *Filename Prefix* edit box.

3. Click the *Update* button, then Click *OK* or press the ENTER key.

2.18 Conclusion

This concludes the Overview of SMS tutorial. The geometry is now ready for the application of model specific boundary conditions.

You can now proceed to which ever model specific tutorial is of interest to you.

If you wish to exit *SMS* at this point:

1. Select the *File* | *Quit* command.

2. Select the *OK* button to confirm.

LESSON 3

Mesh Editing

3.1 Introduction

After a mesh is created there may be some editing required before a numerical analysis can be successfully completed. The production of a final mesh could be time consuming. In this tutorial we will attempt to show briefly the various ways of editing a mesh.

Open the file "tut2.geo" shown in Figure 3-1. To do this:

1. Select *RMA2 | Open Geometry*.

2. Select the file "tut2.geo". If you still have *geometry* open from another tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button or press the ENTER key.

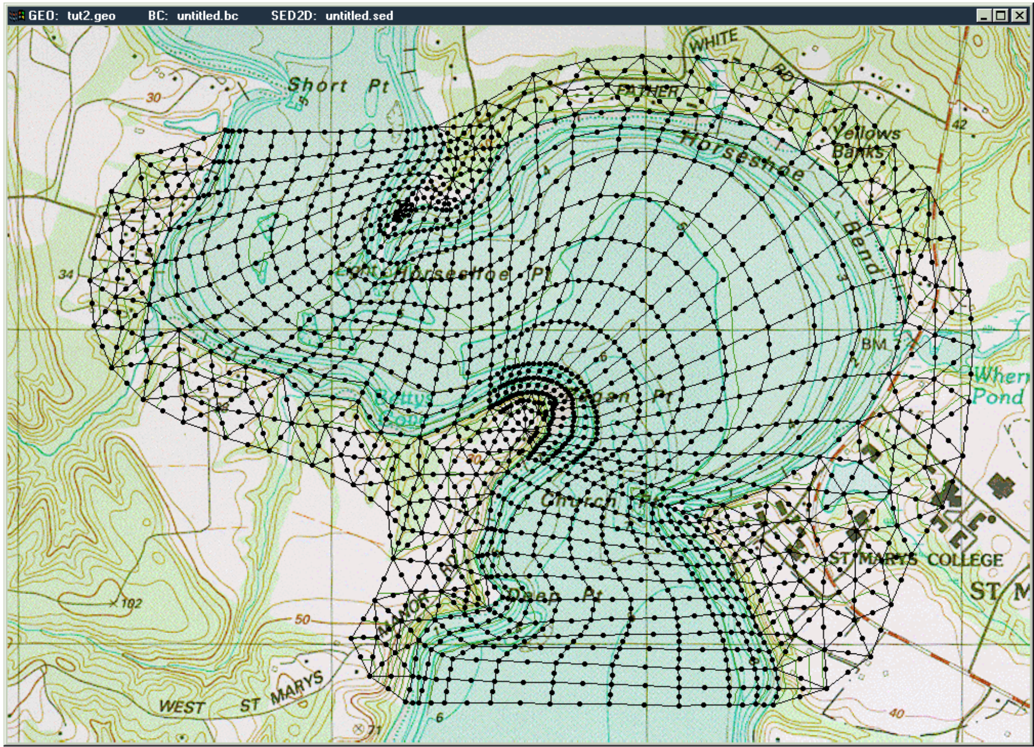
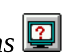


Figure 3-1 The Mesh Contained in tut2.geo

If you do not have the tif file open from Lesson 2 you will need to reopen and register the image (see section 2.4).

3.2 Checking the Mesh

With the rough mesh done it is good to check your mesh and see if it follows the basic topographic information from your scanned map.

1. Open the *Display Options dialog* by selecting the *Display Options*  macro from the *Toolbox*.
2. Turn the Contours toggle on, and the Nodes and Elements toggles off.
3. Click on the Contours option box. This causes the Contour options dialog to appear.
4. Click the *contour between specified range* option found in the contour interval area (middle left area of dialog). Assign a minimum contour value of -32.808 and a maximum contour value of 0.00.
5. Select the *normal linear contours option* from the contour method area (middle right area of dialog).

6. Click the *Color options button* in the other options area (lower right area of dialog).

7. Select the *Use default contour color* radio button.

8. Click on the default color box located under the *Use default contour color* radio button, opening the *line attributes dialog*.

9. Enter a line width of 3, then click *OK* or press the ENTER key

10. Select the *Use range of hues* radio button.

11. Click *OK* or press the ENTER key to exit the *Color Ramp Options* dialog.

12. Click *OK* or press the ENTER key to exit the *Contour Options* dialog.

13. Click the *OK* button or press the ENTER key to exit the D*isplay Options* dialog.

After exiting the *Display Options* dialog, the display is updated with the chosen settings. Now check the mesh to see if the contours follow those contours on the image. If the contours did not follow those of the scanned image to the desired accuracy, then hand editing of elevations could be done, additional scatter data could be used for interpolation or a different interpolation scheme could be employed to generate bathymetry values.

The background image may help in deciding on editing techniques, but in this tutorial it is no longer needed. To increase refresh speed we can simply hide the background image during mesh editing.

1. Choose the *Map Module* icon.

2. Select *Images* | *Hide Image.* This will hide the image from the viewing screen, but it is still loaded and can be used by selecting *Images* | *Show Image* option.

3. Choose the *Mesh Module* icon to return to the mesh module.

3.3 Mesh Quality

The mesh quality option allows you to see possible errors within your mesh so they can be corrected before an analysis is run.

1. Select the *Display Options* macro button.

2. Turn the *Mesh quality* toggle on.

3. Turn contours off, and turn the elements toggle on.

4. Click *OK* or press the ENTER key to exit the *Display options* dialog.

Mesh quality will indicate which elements have smaller than a minimum allowable interior angle, elements with ambiguous gradient, elements with a slope exceeding a maximum slope, elements that are extremely disproportional in size compared to their neighboring elements, and will indicate any concave quadrilaterals if they exist (These will only exist if created by hand in a text editor).

It is best to edit the mesh with mesh quality active. When there are many errors, it is often better to display only one mesh quality indicator at a time. This can be done in the display dialog by selecting the *Options* button next to *Mesh quality*.

The following sections outline methods for mesh editing; some of the editing will be in response to what was shown from the quality mesh option. Most of the edits are performed to illustrate the capabilities of the tools, and are not necessary for this particular mesh case.

3.4 Merging Triangles

The adaptive tesselation process results in polygons composed of all triangles. However quadrilateral elements are often more stable for numerical analysis. The use of quadrilaterals also reduces the number of elements. These factors may result in a mesh which allows solutions to be computed faster. *SMS* provides two options for converting from triangular to quadrilateral elements.

First we will do this for the entire triangulated portion of the mesh.

1. Select *Elements | Merge Triangles*. A dialog will prompt you that no elements have been selected and will ask if you want to merge all triangles, click YES or press the ENTER key.

This process will merge many triangular elements, but will generally not result in a mesh composed entirely of quadrilateral elements. Triangular elements will still remain especially in irregular meshes.

1. Triangles can also be merged by hand. Select the *Split Quads/Merge Triangles* tool. This tool allows you to merge two triangular elements or split a quadrilateral element into two triangles.

2. With this tool active select some grouped triangles that have not already been merged.

3. Simply select a triangular element and if possible it will merge with a neighboring triangular element.

3.5 Splitting Quadrilaterals

In the last section we introduced the *Split Quads/Merge Triangles* tool. Although quadrilateral elements are preferred, there are cases where triangles are required. This is needed to maintain feature boundaries or to aid in mesh quality.

From mesh quality display we see that our mesh contains a few elements with ambiguous gradients. This problem is easily fixed by splitting the element into triangles.

1. Select the *Split Quads/Merge Triangles* tool.

2. Click on the elements outlined in dark green indicating an ambiguous gradient.

This tool will divide the elements into two triangles and thus correct the problem of an ambiguous gradient.

3.6 Swapping Element Edges

Occasionally it is useful to interactively or manually swap the edges of two adjacent triangles. It often provides a simple method to add breaklines and ensure that the edges of the triangular elements honor a geometrical feature or flow pattern that needs to be preserved in the mesh.

1. Select the *Swap Edges* tool from the *Toolbox*.

   With the *Swap Edges* tool selected, clicking on the common edge of two adjacent triangles will cause the edge to be swapped, provided that the trapezoid formed by the two triangles is not concave.

2. Use the Swap Edge tool on the triangles made from the ambiguous elements. Try to swap edges so the elements trend in the same direction as the river. The contours will aid in deciding which orientation is best.

3.7 Select an Area for Editing

In areas of high mesh density, it may be required to zoom in on a particular portion of the mesh. To do this:

1. Open the *Display Options* dialog.

2. Turn the *element numbers* and *nodes* toggles on. Then select *OK* or press the ENTER key.

3. Select the *Zoom*  tool from the *Toolbox* and zoom in on the top right corner of the mesh by dragging a box over the area seen in Figure 3-2.

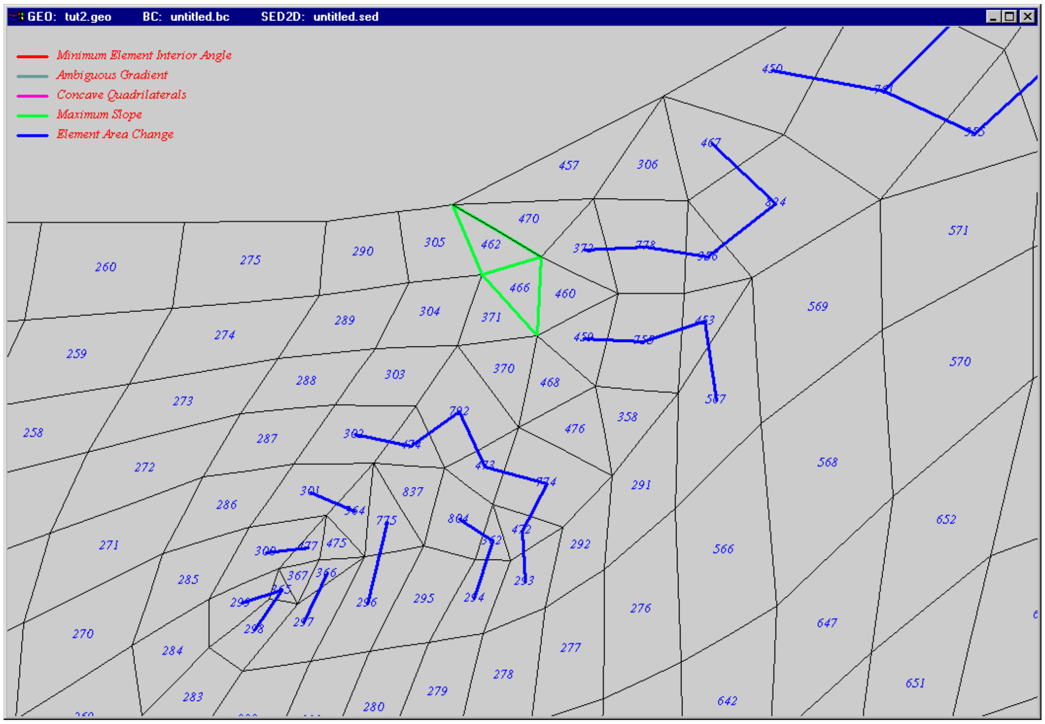


Figure 3-2 Portion of mesh to be edited

This area will be the focus of our editing.

3.8 Inserting and deleting nodes within a mesh

As a mesh is being edited, it may be observed that certain areas of the mesh are too dense or too sparse. This can be rectified by adjusting the parameters in the conceptual model and regenerating the mesh, or nodes can be inserted or deleted by hand using the following steps.

1. Select *Nodes* | *Node Options*.

2. Turn on the options :

   Interpolate function from existing nodes

   Insert nodes into triangulated mesh

   Retriangulate voids when deleting

3. Click *OK* or press the ENTER key.

These options aid in the editing of a mesh when you are editing single nodes. The first option allows for interpolation of elevation when inserting a new node. The second option will allow you to enter a node into the existing mesh and retriangulate the mesh to include the new node. The last option will regenerate the mesh if you delete a node from the mesh.

1. Select the *Select Nodes* tool.

2. Look at the area of elements 470-462-466-460-372 (Figure 3-3). The mesh quality option shows us that the maximum slope is exceeded here. This problem can be fixed by deleting the node common to these elements.

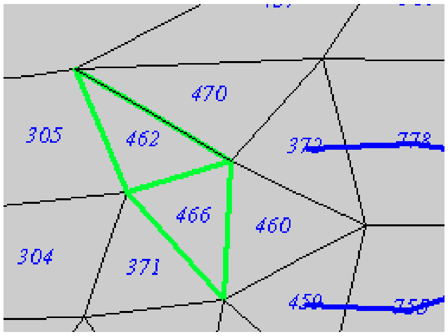


Figure 3-3 Area of node deletion

1. Select the node indicated in Figure 3-3 and press the DELETE key, BACKSPACE key, select *Edit* | *Delete*, or use the *delete* macro. The mesh is retriangulated to fill the void caused by deletion. The new elements now show no problem with maximum slope requirements.

To illustrate how to add a node look at element 260.

1. Select the *Create Nodes* tool from the *Toolbox*.

2. Add a node at the center of this element and observe as the mesh is retriangulated.

3.9 Node Interpolation

Often after a mesh is created it can be seen that more nodes are needed in certain areas. As seen, single nodes can be added by interpolating elevations based on surrounding nodes. Another option is that a set of new nodes can be interpolated between two existing nodes. The locations and elevations of the new nodes are based on an interpolation of the two existing nodes. The user selects the quantity of new nodes, the bias towards a particular node, and whether to interpolate linearly or on an arc.

1. Select the *Select Nodes*  tool and make sure no nodes are selected.

2. Holding the SHIFT key select the two corner nodes on the right side of element 566.

3. Select *Nodes | Node Interp. Opts*.

4. Select the item labeled "*number of intervals in string*" and enter a value of 3 in the text window by this item.

5. We will use the default bias of 1 and default of linear interpolation.

6. Click *OK* or press the ENTER key to execute the interpolation.

Notice that the new nodes were inserted and new elements were created where they crossed existing elements.

3.10 Dragging nodes

When nodes are inserted into an existing mesh they are not always at the exact desired location. Also with the addition of new nodes, pre-existing nodes are no longer in exactly their desired positions. Nodes can be dragged to correct these problems. Also, nodes can be moved to adjust the size and shape of elements surrounding the node.

1. Select *Nodes | Nodes Locked* in order to unlock the nodes. (an * means that the nodes are locked).

2. With the *Select Nodes*  tool selected click on a node a drag it to a new desired location. Notice that the elevation value remains the same and that the shape of the surrounding elements is changed.

3. Using this tool changes the shape of the nodes in the mesh so that the mesh appears more uniform (no sharp turns, transition in element sizes etc.). A good area to edit would be where we added or deleted nodes.

3.11 Refining an element

As has been shown, mesh density can be increased by inserting supplemental nodes. Another option is to refine a specific region of element. This replaces each refined element with four elements. The elevation of the generated nodes is interpolated from the existing nodes.

1. Select the *Select Element*  tool from the *Toolbox*.

2. Select element 570.

3. Select *Elements* | *Refine Elements*.

Note that the chosen element has split into four quadrilateral elements and the mesh density around that element has also increased.

3.12 Renumbering the Elements

As elements are created they are given an element number proceeding that of the last element created. After a mesh is edited the sequence of elements is often scattered and non-uniform. To aid in the analysis of a mesh, elements should be renumbered in an organized way. Renumbering is usually done after all editing and just before analysis.

First we must view a larger portion of the mesh.

1. Select the *Frame* tool from the *Toolbox*.
2. Open the *Display Options* dialog.
3. Turn the E*lement Numbers* and *Mesh Quality* toggles off.
4. Click on the *OK* button or press the ENTER key.

Now a boundary nodestring must be created from which renumbering can originate.

1. Select the *Create Nodestring* tool from the *Toolbox*.
2. Click on the upper far left node of the mesh, then holding the SHIFT key double click on the upper right node of the mesh (see Figure 3-4).

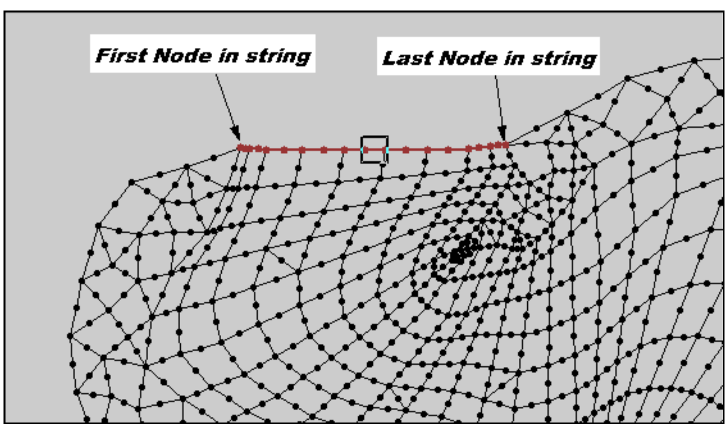


Figure 3-4 Nodestring for renumbering of the mesh.

1. Select the *Select Nodestring* tool and select the recently created nodestring.

2. Select *Elements* | *Renumber*.

3. Choose the option to renumber by *band width*, and select *OK* or press the ENTER key.

SMS will then renumber all of the elements in the mesh based on band width.

3.13 Saving The Geometry

Once the mesh has been edited, the geometry data should be saved to a file. *SMS* can output the geometry into various file formats. See the *SMS* Reference Manual for a description of various file formats. If you are not running a demonstration version, and would like to save your mesh to a GFGEN geometry file for use with *RMA2*:

1. Select *RMA2* | *Save Geometry*.

2. Specify the geo file name. The file "tut2a.geo" has been supplied, but may be overwritten.

3. Click the Save button or press the ENTER key.

Notice that the *Graphics Window* title bar is updated with the name of the geometry file. To save the same mesh to a *FESWMS* geometry file:

1. Select *FESWMS* | Save Simulation.

2. Enter tut2a in the *Filename Prefix* edit box.

3. Click the *Update* button, then Click *OK* or press the ENTER key.

3.14 Conclusion

With these tools, a final mesh can be created tailored to your exact needs. It is seen that hand editing is very time consuming. A carefully defined conceptual model and the use of built-in mesh modification features can make the process much easier and quicker.

This concludes the Mesh Editing tutorial. The geometry is now ready for the application of model specific boundary conditions.

You can now proceed to which ever model specific tutorial is of interest to you.

If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 4

Basic RMA2 Analysis

4.1 INTRODUCTION

This lesson will teach you how to prepare a mesh for analysis and run a solution for *RMA2*. You will be using the file "tut3.geo" and the initial boundary condition file "tut3_0.bc". The geometry has already been created and renumbered following the procedures outlined in Lesson 3.

Open the file "tut3.geo" shown in Figure 4-1. To do this:

1. Select *RMA2| Open Geometry*.

2. Select the file "tut3.geo". If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button or press the ENTER key.

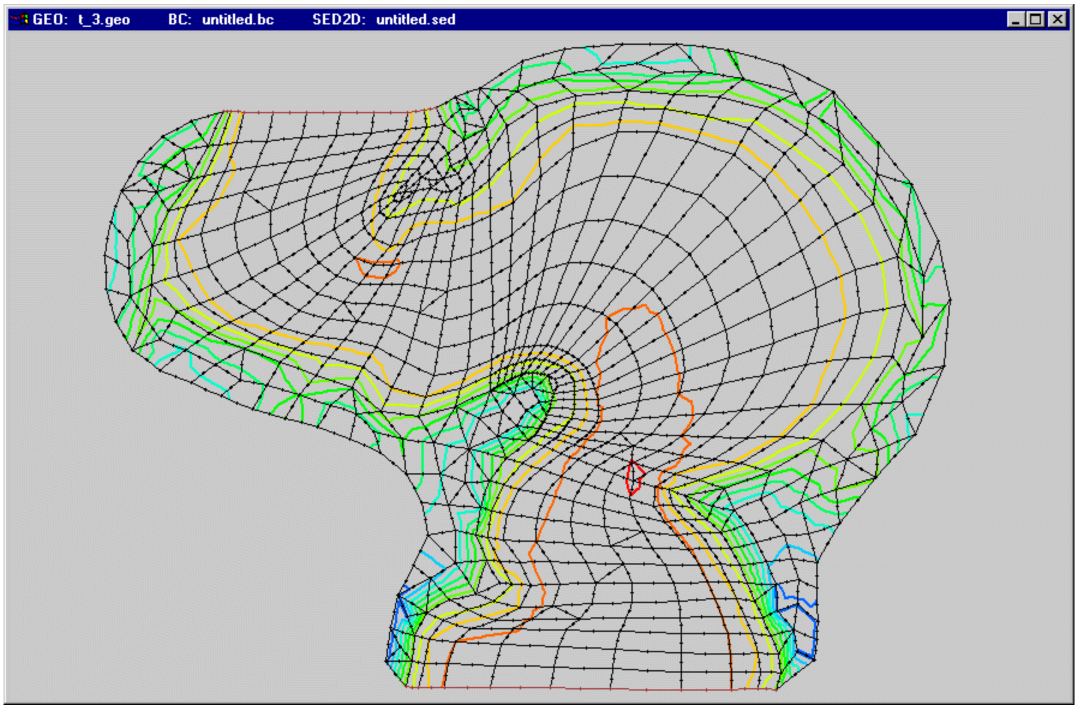


Figure 4-1. The Mesh Contained in tut3.geo.

4.2 CREATING MATERIALS

When this mesh was opened, the elements each contained a material type ID. The materials were created with default parameters which must be changed for this particular mesh. The material properties define how water flows through the element (see the *SMS Reference Manual* for details of what each parameter represents). To edit the material parameters:

1. Select *RMA2| Material Properties*.
2. In the *RMA2 Materials Editor*, select the material labeled *Material 3*. Be sure that its ID number is 3 in the *ID* edit box.
3. Make sure the *Isotropic eddy viscosities* toggle is selected.
4. Enter 25 for *E*. This assigns identical eddy viscosities to *Exx*, *Exy*, *Eyx*, and *Eyy*.
5. Enter 0.025 for *n* and press the TAB key.
6. Now, select the material labeled *Material 4* and check that it has an *ID* of 4.
7. Make sure the *Isotropic eddy viscosities* toggle is selected for this material also.

8. Enter 50 for *E*, then enter 0.075 for *n*.

9. For the materials labeled *Material 1 & 5*, assign an E of 50 and an *n* of 0.04.

10. Click the *Close* button or press the ENTER key.

Each material now has correct parameters associated with it. The materials can be displayed by opening the *Display Options* dialog and turning on the *Materials* toggle. If you desire, turn the materials on to display them.

4.3 DEFINING BOUNDARY CONDITIONS

4.3.1 General Parameters

The geometric mesh is only the first part of the numerical model. We have already defined the material properties associated with the different regions of the mesh. These properties help control how water will flow, and thereby allow us to build a model which matches the physical situation. In addition to the geometry, and the material properties, we must define several other "boundary conditions" and control parameters. These include parameters to specify how to handle situations when the water surface elevation drops and some elements that were part of the model are now essentially dry, or rises and new elements get wet. Other parameters control how the numeric model will operate, including what machine it will be run on, what units the model is specified in, and how many iterations should be made to get the solution to converge. These parameters are set for the *RMA2* analysis model using the *RMA2| RMA-2 Control* command. This command allows access to two dialogs which each contain several control parameters. Information on each parameter can be found in the *SMS* Reference Manual. To simplify the definition of this model, these parameters have been defined in the initial boundary condition file "ld1_init.bc" supplied with the other tutorial files. To open this file:

1. Select *RMA2| Open BC*.

2. Find and select the file "tut3_0.bc", then click the *OK* button or press the ENTER key.

To see what values have been set from this file, select *RMA2| RMA-2 Control*, and examine the setting in the *Global BC Control* and the *Optional BC Control* dialogs.

4.3.2 Defining Steady State Flow and Head

Steady state boundary conditions, as the name implies, do not change with time. For this tutorial, flow and water surface elevation will be defined along nodestrings at the open boundaries of the mesh. An open boundary is a boundary where flow enters or exits from the mesh. Generally for *RMA2*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries.

Two boundary strings must be created along the open boundaries. These boundaries are highlighted in Figure 4-2. The top boundary string was created in Lesson 3, but the lower nodestring still needs to be defined.

1. Select the *Create Nodestrings* tool  from the *Toolbox*.

2. Start the string by clicking on the lower left corner node.

3. Hold down the SHIFT key and click on the lower right corner node of the outflow boundary. *SMS* will automatically select all nodes between the two. (The SHIFT key causes the nodestring to progress through the grid by the largest steps possible. Another option would be to hold the CONTROL key which causes the nodestring to proceed counter clockwise around the border of the mesh or the SHIFT and CONTROL keys together which causes the nodestring to proceed clockwise around the border.)

4. Press the ENTER key, or double click on the last node in the nodestring to terminate the string.

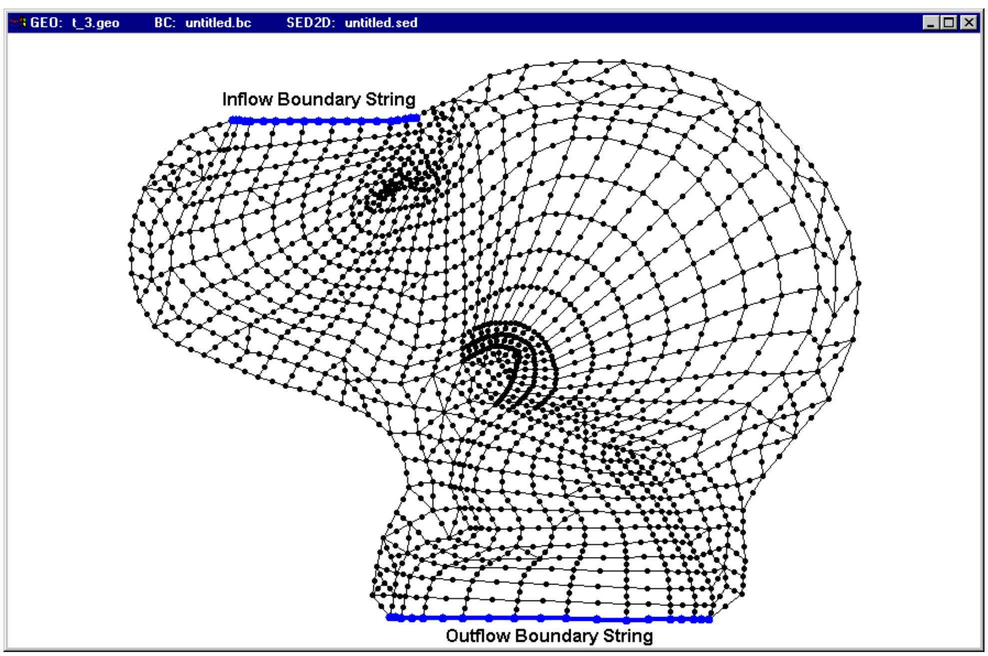


Figure 4-2. Position of the Boundary Strings in tut3.geo.

After the nodestrings are created, boundary data may be assigned to each. To assign the steady state flow conditions:

1. Click on the *Select Nodestrings* tool  in the *Toolbox*.

2. Select the top nodestring by clicking in the box at the center of the string.

3. Select *RMA2| Assign BC* and click the *Flow BC* and *Constant* radio buttons.

4. Assign a value of 50,000 (cfs) in the *Flowrate* edit box and click on the *Perp to Boundary* button to calculate the angle.

5. Click *OK* or press the ENTER key.

The selected nodestring is now defined as a flow string, and an arrow appears indicating flow direction. Now, assign the steady state water surface elevation (head) to the downstream open boundary. To do this:

1. Select the bottom nodestring.

2. Select *RMA2| Assign BC* and click the *Head BC* radio and *Constant* radio buttons.

3. Assign a value of 60 (feet) in the *Elevation* edit box.

4. Click the *OK* button or press the ENTER key.

Now a head symbol appears at the center of the nodestring, indicating that it is a water surface elevation string.

4.4 SAVING THE BOUNDARY CONDITIONS

RMA2 uses both the geometry file and the boundary condition file to run an analysis. You must save the boundary conditions that you just created. To do this:

1. Select *RMA2| Save BC.*

2. Enter the filename "tut3.bc" and click the *Save* button or press the ENTER key.

The boundary conditions you specified are now saved. Before continuing with the analysis, check the model for completeness. To do this:

1. Select *RMA2| Model Check*.

2. Click the *Run Check* button.

The *RMA2* model checker will report one possible error. The warning is that the mesh has not been renumbered during this session. This is not important because it was renumbered before it was saved. Errors reported by the model checker usually deal with the stability of a mesh.

Often when a mesh will contain many dry elements, the solution must be reached through a series of *spin down* steps. The first step is at a low flow and a high water surface elevation that will allow coverage of all elements. This is what we have done for this particular mesh. It is often called a *cold start* and can be used as a starting point for further mesh analysis. From this point a series of *spin down* solutions can be made or a dynamic solution that steps down to the desired head can be created.

Using a file that has been *spun down* is one means of enhancing the stability of a mesh. *RMA2* can use this variant boundary condition as a *hotstart* for our steady state boundary conditions. This procedure is shown in Lesson 5 of the tutorials.

4.5 DEFINING TIME DEPENDENT FLOW AND HEAD

Time dependent flow conditions are generally used to model a changing tide (time dependent water surface elevation) or changing flow in a stream (time dependent flow). However, as discussed in the previous section, time dependent boundary conditions may also be used to increase the stability of the model. For instructions on how to set time dependent conditions refer to Lesson 5.

4.6 USING GFGEN

GFGEN is a processor program used to convert the ASCII geometry file created by *SMS* into a binary format for use by *RMA2*. *GFGEN* only requires the ASCII geometry file. To run *GFGEN*:

1. Path to the directory containing the geometry file.

2. Type gfgen (or gfgv430) to execute GFGEN.

3. Respond to the prompts that appear as follows:
   ENTER GFGEN RUN CONTROL INPUT FILE NAME
   tut3.geo
   ENTER FULL PRINT OUTPUT FILE NAME
   tut3.ot1
   ENTER THE BINARY OUTPUT GEOMETRY FILE NAME
   tut3.bin

When *GFGEN* has completed the conversion, a beep will sound. The file "tut3.bin" is used as input to *RMA2*, along with the boundary condition file that you created in *SMS*.

4.7 USING RMA2

RMA2 is the analysis program which computes 2D flow solutions (water surface elevation, water depth, and velocity) at each node. *RMA2* requires the binary geometry file created by GFGEN and the ASCII boundary condition file created by *SMS*. Depending on the boundary condition file, *RMA2* will calculate either a steady state solution with only a single time step, or a dynamic solution with multiple time steps. Steady state and dynamic solutions are computed independent of each other. For the mesh used in this tutorial, we created a steady state boundary condition file.

Note that the boundary condition file for the steady state analysis includes a $L card with a flag set to create a *hotstart* file. This specification is made in the *Global BC Control* dialog for *RMA2*. If this *hotstart* option is not set, *RMA2* will not ask for the output *hotstart* file.

To run *RMA2* with the steady state boundary conditions:

1. Path to the directory containing the boundary condition file and the output from GFGEN.

2. Type “*RMA2*v435" to execute the program.

3. Respond to the prompts that appear as follows:
    ENTER RUN CONTROL INPUT FILE NAME
    tut3.bc
    ENTER FULL PRINT OUTPUT FILE NAME
    tut3.ot3
    ENTER INPUT GEOMETRY FILE (BINARY)
    tut3.bin
    ENTER FINAL *RMA2* RESULTS FILE (BINARY)
    tut3.sol

RMA2 can take between two to five minutes to run this solution, depending on the speed of your computer. When *RMA2* has completed the calculations, a beep will sound. The file "tut3.sol" is the solution file containing flow and head data at each node. The file “tut3.hot” is a hotstart file which can be used in subsequent mesh solutions.

4.8 CONCLUSION

This concludes the Basic RMA2 Analysis tutorial. Viewing the results of the *RMA2* analysis is discussed in Lesson 9, and is based on the data from next lesson of the tutorials.

If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 5

Advanced RMA2 Analysis

5.1 INTRODUCTION

This lesson will teach you how to prepare a geometry for analysis and run a solution for *RMA2*. You will be using the file "ld1.geo" and the initial boundary condition files "ld1_init.bc" and "ld0_init.bc". The geometry has already been created and renumbered following the procedures outlined in Lessons 2 & 3.

Open the file "ld1.geo" shown in Figure 5-1. To do this:

1. Select *RMA2| Open Geometry*.

2. Select the file "ld1.geo". If you still have *tut1* open from the previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button or press the ENTER key.

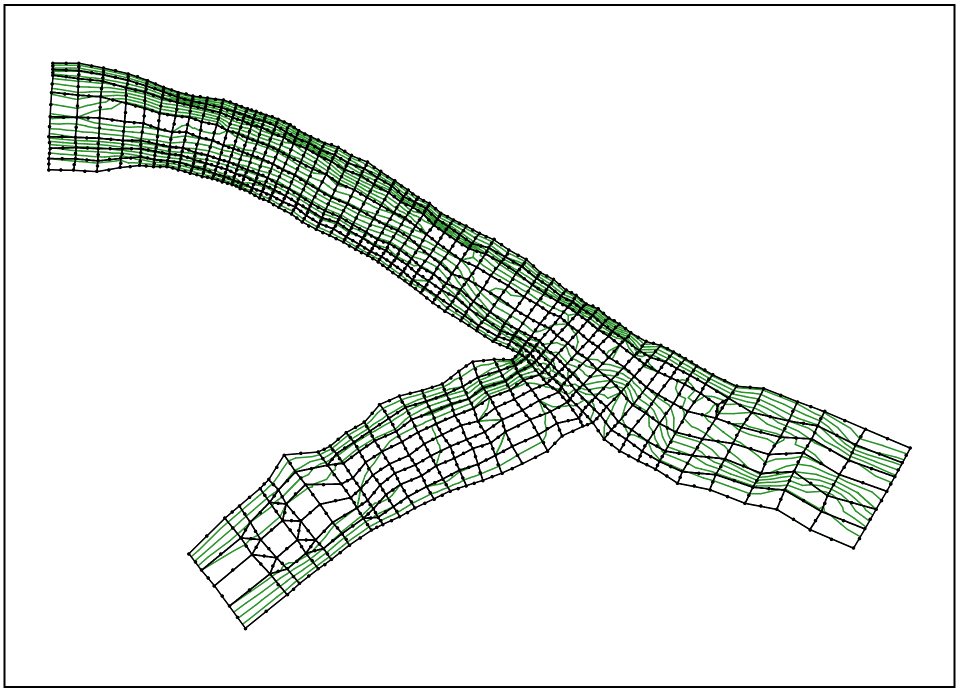


Figure 5-1. The Mesh Contained in ld1.geo.

5.2 CREATING MATERIALS

When this mesh was opened, the elements each contained a material type ID. The materials were created with default parameters which must be changed for this particular mesh. The material properties define how water flows through the element (see the *SMS Reference Manual* for details of what each parameter represents). To edit the material parameters:

1. Select *RMA2| Material Properties*.

2. In the *RMA2 Materials Editor*, select the material labeled *Material 1*. Be sure that its ID is number 1 in the *ID* edit box.

3. Make sure the *Isotropic eddy viscosities* toggle is selected.

4. Enter 25 for *E*. This assigns identical eddy viscosities to *Exx*, *Exy*, *Eyx*, and *Eyy*.

5. Enter 0.025 for *n* and press the TAB key.

6. Now, select the material labeled *Material 2* and check that it has an *ID* of 2.

7. Make sure the *Isotropic eddy viscosities* toggle is selected for this material too.

8. Enter 50 for *E*, then enter 0.04 for *n*.

9. Click the *Close* button or press the ENTER key.

Each material now has correct parameters associated with it. The materials can be displayed by opening the *Display Options* dialog and turning on the *Materials* option. If you desire, turn the materials on to display them.

5.3 DEFINING BOUNDARY CONDITIONS

5.3.1 General Parameters

The geometric mesh is only the first part of the numerical model. We have already defined the material properties associated with the different regions of the mesh. These properties help control how water will flow, and thereby allow us to build a model which matches the physical situation. In addition to the geometry, and the material properties, we must define several other "boundary conditions" and control parameters. These include parameters to specify how to handle situations when the water surface elevation drops and some elements that were part of the model are now essentially dry, or rises and new elements get wet. Other parameters control how the numeric model will operate, including what machine it will be run on, what units the model is specified in, and how many iterations should be made to get the solution to converge. These parameters are set for the *RMA2* analysis model using the *RMA2| RMA-2 Control* command. This command allows access to two dialogs which each contain several control parameters. Information on each parameter can be found in the *SMS Reference* Manual. To simplify the definition of this model, these parameters have been defined in the initial boundary condition file "ld1_init.bc" supplied with the other tutorial files. To open this file:

1. Select *RMA2| Open BC.*

2. Find and select the file "ld1_init.bc", then click the *OK* button or press the ENTER key.

To see what values have been set from this file, select *RMA2| RMA-2 Control*, and examine the setting in the *Global BC Control* and the *Optional BC Control* dialogs.

5.3.2 Defining Steady State Flow and Head

Steady state boundary conditions, as the name implies, do not change with time. For this tutorial, flow and water surface elevation will be defined along nodestrings at the open boundaries of the mesh. An open boundary is a boundary where flow enters or exits from the mesh. Generally for *RMA2*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries.

Three boundary strings must be created along the open boundaries. These boundaries are highlighted in Figure 5-2. To create a nodestring across the upper inflow boundary, use these steps:

1. Select the *Create Nodestrings* tool  from the *Toolbox*.

2. Start the string by clicking on the upper left corner node.

3. Hold down the SHIFT key and click on the bottom corner node of the upper inflow boundary. *SMS* will automatically select all nodes between the two. (The SHIFT key causes the nodestring to progress through the grid by the largest steps possible. Another option would be to hold the CONTROL key which causes the nodestring to proceed counter clockwise around the border of the mesh or the SHIFT and CONTROL keys together which causes the nodestring to proceed clockwise around the border.)

4. Press the ENTER key, or double click on the last node in the nodestring to terminate the string.

In a similar fashion, create nodestrings at the other two open boundaries.

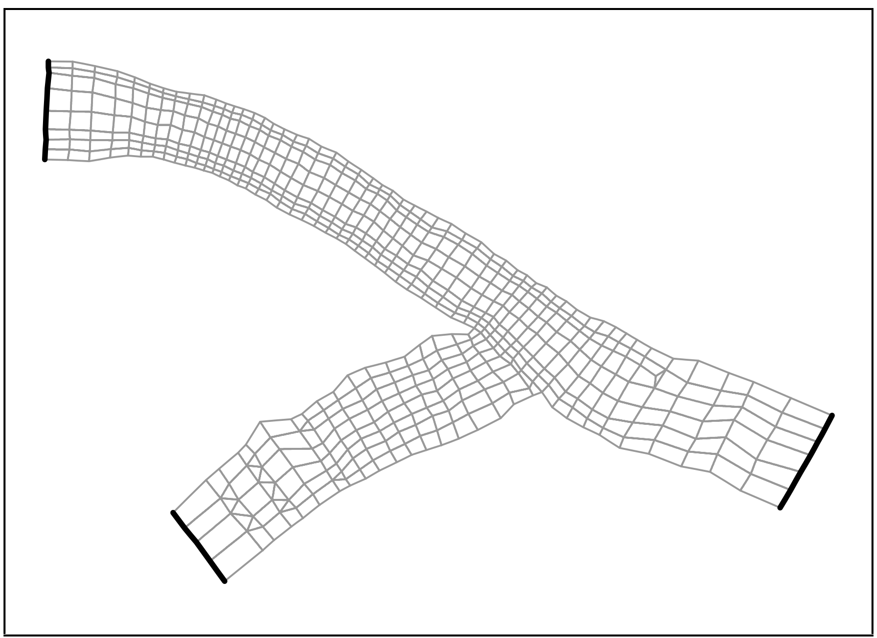


Figure 5-2. Position of the Boundary Strings in ld1.geo.

After the nodestrings are created, boundary data may be assigned to each. To assign the steady state flow conditions:

1. Click on the *Select Nodestrings* tool  in the *Toolbox*.

2. Select the top-left nodestring by clicking in the box at the center of the string.

3. Select *RMA2| Assign BC* and click the *Flow BC* and *Steady State* radio buttons.

4. Assign a value of 55000 (cfs) in the *Flowrate* edit box and click on the *Perp to Boundary* button for the angle.

5. Click *OK* or press the ENTER key.

The selected nodestring is now defined as a flow string, and an arrow appears indicating flow direction. In the same manner as above, assign a value of 580 (cfs) of flow to the bottom left nodestring, also using the *Perp to Boundary* angle option. Now, assign the steady state water surface elevation (head) to the downstream open boundary. To do this:

1. Select the right nodestring.

2. Select *RMA2| Assign BC* and click the *Head BC* radio button.

3. Assign a value of 33.3 (feet) in the *Elevation* edit box.

4. Click the *OK* button or press the ENTER key.

This time, a head symbol appears at the center of the nodestring, indicating that it is a water surface elevation string.

5.4 SAVING THE BOUNDARY CONDITIONS

RMA2 uses both the geometry file and the boundary condition file to run an analysis. You must save the boundary conditions that you just created. To do this:

1. Select *RMA2| Save BC*

2. Enter the filename "ld1.bc" and click the *Save* button or press the ENTER key.

The boundary conditions you specified are now saved. Before continuing with the analysis, check the model for completeness. To do this:

1. Select *RMA2| Model Check*.

2. Click the Run Check button.

The *RMA2* model checker will report two possible errors. The first warning is that the mesh has not been renumbered during this session. This is not important because it was renumbered before it was saved. The second warning is that the water surface elevation at the downstream open boundary is lower than some of the nodes in the mesh. This is a warning because *RMA2* only supports subcritical flow, and a water surface elevation that is below the elevation of some nodes indicates a potentially large head difference which may result in supercritical flow. Supercritical flow will not always be the result in such a case, but a warning is given to indicate that it is a possibility. Therefore, steps should be taken to try to insure numerical stability during the analysis of the mesh.

One means of enhancing the stability is to use another boundary condition file that starts with a higher water surface elevation, and slowly lowers the head to what we have specified. *RMA2* can use the results of an analysis using this variant boundary condition as a *hotstart* for our steady state boundary conditions. Therefore we need to define a time dependent set of boundary conditions.

5.5 DEFINING TIME DEPENDENT FLOW AND HEAD

Time dependent flow conditions are generally used to model a changing tide (time dependent water surface elevation) or changing flow in a stream (time dependent flow). However, as discussed in the previous section, time dependent boundary conditions may also be used to increase the stability of the model. For this tutorial, we will import three time series curves, one for each boundary string. (See the *SMS Reference Manual* for more information on creating time series curves.) *RMA2* is a subcritical flow model. If the slope of the mesh is too great, supercritical flow will result. If the flow is approaching supercritical, it may be necessary to help *RMA2* maintain numerical stability. One way to do this is to start with a higher water surface elevation and generate a safe subcritical flow solution, and use that solution as a starting point for a lower water surface elevation. This process can be repeated until the water surface elevation is at the desired level. This can be done one step at a time using hot-start files, however, a simpler solution is to approximate the process using a dynamic water surface elevation as a boundary condition. As with the steady state boundary conditions, some initial parameters contained in the file "ld0_init.bc" must be set. To open this file:

1. Select *RMA2| Open BC.*

2. Find and select the file "ld0_init.bc". If current boundary condition data is present, you will be prompted to remove all such data. At the prompt, click the *Yes* button or hit the ENTER key.

By examining these parameters, we see that the option to save an output *hotstart* file has been enabled. This means that the results from this solution will be in a form that may be used as a starting point for the steady state problem. With these parameters are set, we can assign flow and head. To do this, we will import curves that were previously created for this purpose. To assign the time dependent flow:

1. Select the top left nodestring that you previously assigned steady state flow.

2. Select *RMA2| Assign BC.*

3. Click the *Transient* radio button, then the *Define Curve* button.

4. The *XY Series Editor* that will appear. Click on the *Import* button.

5. Select the file "flow1_ld.xys" and click *OPEN*.

6. A straight line appears in the time series window.

7. Click *OK* or hit the ENTER key to exit the *XY Series Editor*. The time series curve will be displayed in the window next to the *Define Curve* button.

8. Click the *OK* button or press the ENTER key.

The nodestring now has dynamic flow associated with it. In the same manner, select the bottom left nodestring and import the curve "flow2_ld.xys", (which is also a straight line). This assigns this curve to the lower inflow boundary. Now select the right nodestring and import and assign the curve "head_ld.xys" to it. The head curve is a parabolic shape. Now save the time dependent boundary conditions as described above, using "ld0.bc" as the file name. Running the *RMA2 Model Checker* for this problem should not generate any warnings.

Once a mesh has been created and boundary conditions assigned, it is ready for analysis. To execute the analysis programs, you must bring up another terminal window if you are on a UNIX platform, or a DOS prompt if you are on a MS Windows platform.

5.6 USING GFGEN

GFGEN is a processor program used to convert the ASCII geometry file created by *SMS* into a binary format for use by *RMA2*. *GFGEN* only requires the ASCII geometry file. To run *GFGEN*:

1. Path to the directory containing the geometry file.

2. Type gfgen (or gfgv430) to execute GFGEN.

3. Respond to the prompts that appear as follows:

   ENTER GFGEN RUN CONTROL INPUT FILE NAME
   ld1.geo
   ENTER FULL PRINT OUTPUT FILE NAME
   ld1.ot1
   ENTER THE BINARY OUTPUT GEOMETRY FILE NAME
   ld1.bin

When *GFGEN* has completed the conversion, a beep will sound. The file "ld1.bin" is used as input to *RMA2*, along with the boundary condition file that you created in *SMS*.

5.7 USING RMA2

RMA2 is the analysis program which computes 2D flow solutions (water surface elevation, water depth, and velocity) at each node. *RMA2* requires the binary geometry file created by GFGEN and the ASCII boundary condition file created by *SMS*. Depending on the boundary condition file, *RMA2* will calculate either a steady state solution with only a single time step, or a dynamic solution with multiple time steps. Steady state and dynamic solutions are computed independent of each other. For the mesh used in this tutorial, we created a dynamic boundary condition file that starts with a high water surface elevation for a stable initial solution and lowers the elevation to a lower elevation which is less stable. This dynamic boundary condition file should be used first. The dynamic boundary condition file specifies in the $L card that *RMA2* should create a *restart* or *hotstart* file for the steady state solution. Therefore, *RMA2* will prompt the user for the name of this file. If the flag in the $L card is not set, *RMA2* will not ask for a *hotstart* file name. To run *RMA2* with the dynamic boundary conditions:

1. Path to the directory containing the boundary condition file and the output from GFGEN.

2. Type "*RMA2*v435" to execute the program.

3. Respond to the prompts that appear as follows:

   ENTER RUN CONTROL INPUT FILE NAME
   ld0.bc
   ENTER FULL PRINT OUTPUT FILE NAME
   ld1.ot2
   ENTER INPUT GEOMETRY FILE (BINARY)
   ld1.bin
   ENTER BINARY OUTPUT RESTART/HOTSTART FILE (BINARY)
   ld0.rsr (*NOTE: Not every model requires a restart/hotstart file).*
   ENTER FINAL *RMA2* RESULTS FILE (BINARY)
   ld0.sol.

RMA2 can take between twenty to forty minutes to run this dynamic solution, depending on the speed of your computer. When *RMA2* has completed the calculations, a beep will sound. The file "ld0.sol" is the dynamic solution file containing flow and head data at each node. The file "ld0.rsr" is the restart file that stores the current velocities, water surface elevations, etc. to be used as starting values in the subsequent steady state solution. Now that the *hotstart* file has been created, the steady state solution can be run. Note that the boundary condition file for the steady state analysis includes a $L card with a flag set to use a *hotstart* file. This specification is made in the *Global BC Control* dialog for *RMA2*. If this *hotstart* option is not set, *RMA2* will not ask for the output *hotstart* file from the above run, and the problem would be unstable. To run *RMA2* with the steady state boundary conditions:

1. Type "*RMA2*v435" to execute the program.

2. Respond to the prompts that appear as follows:

   ENTER RUN CONTROL INPUT FILE NAME
   ld1.bc
   ENTER FULL PRINT OUTPUT FILE NAME
   ld1.ot3
   ENTER INPUT GEOMETRY FILE (BINARY)
   ld1.bin
   ENTER BINARY INPUT RESTART/HOTSTART FILE (BINARY)
   ld0.rsr (*NOTE: Not every model requires a restart/hotstart file)*.
   ENTER FINAL *RMA2* RESULTS FILE (BINARY)
   ld1.sol

This steady state solution only takes a couple of minutes to run. When *RMA2* has completed the calculations, a beep will sound. The file "ld1.sol" is the final solution and contains steady state flow and head data at each node.

5.8 CONCLUSION

This concludes the Advanced RMA2 Analysis tutorial. We suggest that you now proceed to the tutorial 2D Post-Processing included in Lesson 10 to view the result of this *RMA2* analysis. The user may also be interested in proceeding to Lesson 6 on SED2D-WES Analysis, which uses the results of this tutorial.

If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 6

SED2D-WES Analysis

6.1 INTRODUCTION

This lesson will teach you how to prepare a geometry for analysis and run a solution for *SED2D-WES*. If you have not yet completed Lesson 5 on *RMA2*, you should do so now to understand the process of creating the input files for *RMA2* and running an analysis to generate a solution file. The required solution files are included with evaluation copies of *SMS*. *SED2D-WES* uses the 2D flow solutions from *RMA2*. You will need the geometry file "ld1.geo", boundary condition file "ld1.bc", and the solution file "ld1.sol", that was obtained by running *RMA2* on the ld1 mesh in tutorial 5.

6.2 PREPARING FOR SED2D-WES

SED2D-WES can only be run after having initially run *RMA2*. This is because *SED2D-WES* uses the flow solutions computed by *RMA2* to compute where the sediment will go as it flows through the mesh. For this reason, the entire *SED2D-WES* menu is dimmed out until a valid *RMA2* boundary condition file has been read in. To enable the *SED2D-WES* menu:

1. Open the "ld1.geo" *GFGEN* geometry file by selecting *RMA2| Open Geometry*.

2. Find and select the file "ld1.geo".

3. Open the "ld1.bc" boundary condition file by selecting *RMA2| Open BC*.

4. If you have modified the currently loaded boundary conditions since they were last saved, you will be asked if you wish to continue without saving. If this prompt occurs, click the *Yes* button.

5. Find and select the file "ld1.bc".

After *SMS* successfully reads in the boundary condition file, the *SED2D-WES* menu will be enabled.

6.3 SETTING UP THE GLOBAL PARAMETERS

The first thing to prepare for a *SED2D-WES* run is to define the bed type of the mesh. Either a sand or clay bed can be specified, but not both. To specify the global parameters of the flow bed:

1. Choose *SED2D-WES| Global Parameters*.

2. Make sure a *Sand* bed type is selected in the type radio group and then click on the *Set Up Bed* button.

3. Input the following values for the sand bed parameters:

    - 0.07 for *min* and *max grain size*;
    - 2.65 for *Specific gravity*
    - 0.67 for *Grain shape factor*
    - 1.0 for *Characteristic deposition length factor*
    - 10.0 for *Characteristic erosion length factor*
    - 0.0 for *thickness*
    - 0.07 for *Sand grain roughness*
    - 0.07 for *Sand grain size*

4. Click *OK* or press the ENTER key.

5. Enter these values for the other parameters

    - 25.0 for xx and yy diffusion coefficients
    - 0.5 for Initial Concentration
    - 0.0038 for Initial Settling Velocity

6. Click *OK* or press the ENTER key.

Now that the bed parameters have been defined globally, the next step is to define the model controls. These are parameters that are used to verify the model. To define the model controls for this mesh:

1. Choose *SED2D-WES| Model Control*.

2. In the *Hydraulic Bed Shear Stress* area, select *Manning's Equation*. This means that bed shear stresses will be computed using Manning's equation.

3. Insert a value of 0.66 for the *Crank-Nicholson Theta*.

4. In the *Time Controls* area, enter the following values:

   - 2.0 for the *Time step*
   - 24.0 for *Simulation time*
   - 12 for *Number of cycles*
   - 24 for *Repeat increment*
   - 1 for *Output frequency*

5. Make sure the *Test Convergence* option is turned off, and enter a value of 1000 for *Water Density*.

6. Click *OK* or press ENTER key to exit the *Model Control* dialog.

When running the solution, the output is printed to a file. Since this mesh is quite large, the printed output file can become large. To change the print settings:

1. Choose *SED2D-WES| Print Control*.

2. Turn the *Print Echo Control* option *ON* and enter a value of 20.

3. Make sure *Subroutine Trace* printing is *OFF*.

4. Click *OK* or press ENTER key to exit the *Print Control* dialog.

6.4 CREATING INITIAL SEDIMENT CONCENTRATION

After all of the global parameters have been defined for the entire mesh, specific parameters can be defined for groups of nodes, elements, nodestrings, or material types. These parameters are called *Local Parameters* and *BC Concentrations*. Local parameters can be defined anywhere within the mesh, but boundary concentrations for sediment load coming into the mesh may be defined only at inflow boundaries.

In this case, you will assign an initial sediment concentrations to the top left nodestring that has a specified flow assigned to it. To assign the initial concentrations:

1. Select the top left nodestring that exists as a flow string. (Don't forget to click on the *Select Nodestring*  tool first).
2. Choose *SED2D-WES| BC Concentrations*.
3. Make sure the *Steady State* button is selected and enter a value of 0.50. This means that a sediment concentration of 0.5 passes each node of the nodestring at each time step.
4. Click *OK* or press ENTER key.

A symbol will appear at each node of the nodestring indicating that *SED2D-WES* data was assigned to them. The default symbol is a blue box, but this may be changed by the user using the *SED2D-WES Display Options* dialog. A zoomed-in view of the nodestring with the initial concentrations defined is in Figure 6-1.

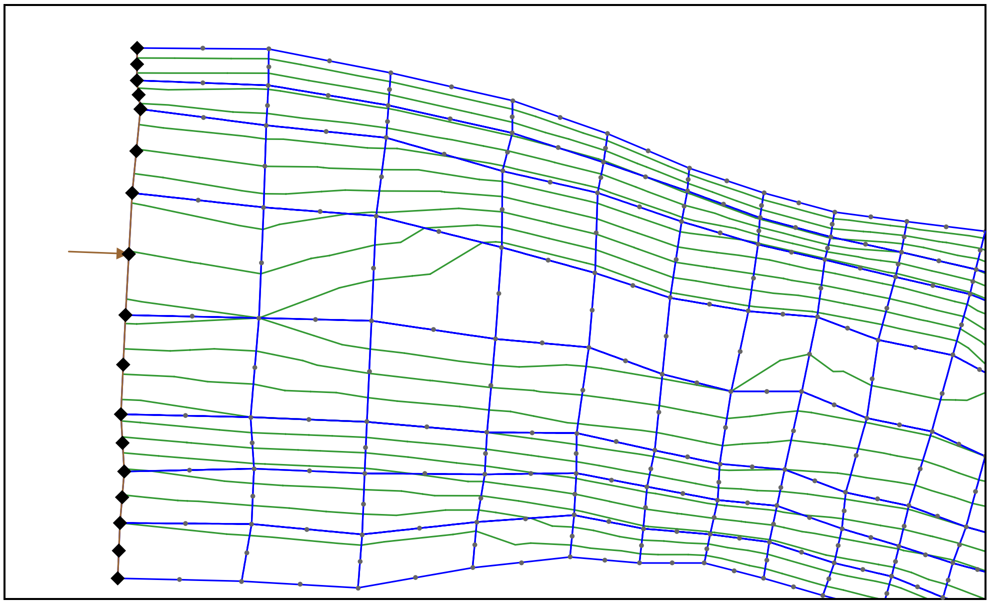


Figure 6-1. Zoomed View of Nodes with SED2D-WES *Boundary Conditions.*

6.5 SAVING THE DATA

Now that the initial concentrations have been defined, you need to save the data before you can run the analysis. To save the data to a file:

1. Select *SED2D-WES | Save Simulation*.

2. Enter the name "*ld1.sed"* and click the *OK* button or press the ENTER key.

6.6 USING SED2D-WES

You are now ready to run an analysis. To run *SED2D-WES*, you need the "ld1.bin" geometry file created with GFGEN, the "ld1.sol" file created with *RMA2*, and the ld1.sed file you just saved. To run *SED2D-WES*:

1. Path to the directory containing the files mentioned above.

2. Type *SED2D* (or *SED2D*v11) to execute the program.

3. Respond to the prompts that appear as follows:

   ENTER RUN CONTROL INPUT FILE NAME
   ld1.sed
   ENTER FULL PRINT OUTPUT FILE NAME
   ld1.ot3
   ENTER INPUT GEOMETRY FILE FROM GFGEN (BINARY)
   ld1.bin
   ENTER INPUT *RMA2* HYDRODYNAMIC FILE (BINARY)
   ld1.sol
   ENTER OUTPUT CONCENTRATION/DELBED FILE (BINARY)
   ld1_dbed.sol
   ENTER OUTPUT BED STRUCTURE (BINARY)
   ld1_obed.sol
   ENTER OUTPUT GEOMETRY CONTAINING NEW BATHYMITRY (ASCII)
   ld2.geo

SED2D-WES will take one or two minutes to finish the solution. The "ld1_dbed.sol" file contains the solution data needed. The "ld1_obed.sol" file is a file reserved for future development in *SED2D-WES*, and is not used right now. The "ld2.geo" contains a new bathymetry, or mesh, in which the nodal elevations have been modified to represent the deposition or scour that has taken place. This new geometry can then be run through *GFGEN*, *RMA2*, and then again through *SED2D-WES* so that each run is updated with the new solutions. *SED2D-WES* was meant to be used in an iterative manner with *RMA2*.

6.7 CONCLUSION

This concludes the SED2D-WES Analysis tutorial. We suggest that you now proceed to the 2D Post-Processing tutorial included in Lesson 10 to learn how to view the result of this analysis.

If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 7

Basic FESWMS Analysis

7.1 INTRODUCTION

This lesson will teach you how to prepare a mesh for a *FESWMS* solution run. You will be using the file "tut7.fil". This file has been created in a manner consistent with that which has already been outlined in Lesson 2. *FESWMS* supports both 8-noded quadrilateral elements and 9-noded quadrilateral elements. Elements with 8-nodes have been used in this tutorial. The model has also been renumbered. Only the band width renumbering method should be used for *FESWMS* models since the band width method attempts to account for equations used by *FESWMS*. A *FESWMS .fil* file is a type of super file. It contains a list of filenames that may be used by *FESWMS*. The actual input data is stored in the files named in the super file.

Open the file "tut7.fil" which causes *SMS* to read the data shown in Figure 7-1. To do this:

1. Select *FESWMS| Open Simulation*.

2. Select the file "tut7.fil" and click the *OPEN* button or press the ENTER key. If a mesh is open, you will be warned that all existing data will be deleted. If this happens, click the *OK* button or press the ENTER key.

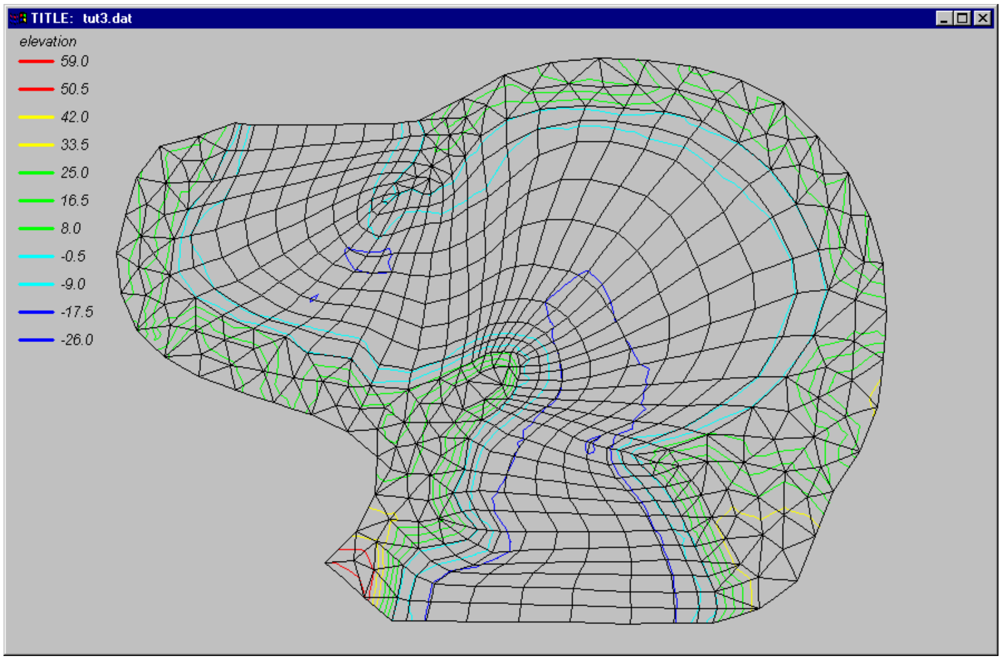


Figure 7-1. The mesh contained in tut7.fil.

7.2 CREATING MATERIALS

When the mesh was opened, the elements each contained one of four material type ID's. The materials were created, but contain default values which must be changed (see the *FESWMS* chapter of the *SMS Reference Manual* for details of what each material parameter represents). To change the material values:

1. Select *FESWMS| Material Properties*.
2. In the upper right corner of the *FESWMS Material Editor* dialog, a graphical image shows what the Manning coefficients at different depths are.
3. Select the material named *Material 3*, and be sure that the *ID* is 3.
4. Enter the following values:
   - 0.025 for both *Manning coefficients (n1 and n2).*
   - 2.0 for *Depth 1*
   - 3.0 for *Depth 2*
   - 0.0 for *Chezy*
   - 20.0 for *Vo*

- 0.6 for *Cu*

5. Now, select the material labeled *Material 4* and check that it has an *ID* of 4.

6. Enter the same values for *Material 4* that were entered for *Material 3* with the exception of the *Manning coefficients* which should be 0.075.

7. For *Materials* 1 and 5, enter the same values with the exception that both *Manning coefficients* are 0.04.

8. Click the *Close* button or press the ENTER key.

You have just assigned values for the four materials in this mesh.

Optional: The materials can be displayed by opening the *Display Options* dialog and clicking the toggle box next to the word *Materials*. See the *SMS Reference Manual* for more about changing the display of materials. Before continuing with this tutorial, turn the *Materials* option *OFF* in the *Display Options* dialog if you had previously turned it *ON*.

7.3 ASSIGNING BOUNDARY CONDITIONS

Boundary conditions define existing parameters, such as flow and water surface elevation. Without proper boundary conditions, a model will not be stable. Further, the values of the boundary conditions, coupled with the geometric attributes, control the solution. After a model is constructed and analyzed, it should be verified. This means that the numerical results must match the physical conditions. To verify a model, we must first be sure that the mesh correctly represents the shape of the region being modeled. We must then be sure that the boundary conditions are accurate measures of the physical conditions. Finally, we can adjust the material parameters to change the performance of the model.

Often to increase the stability of a mesh, the solutions are *spun down* until the desired solution is achieved. The initial solution is called a *cold start* file and is given a high water surface elevation so that there are no dry elements within the mesh. This solution is then used as a initial conditions (*hotstart*) file for subsequent solutions, each subsequent solution is made with a lower head until the desired solution is achieved. In this tutorial we will create a *cold start* solution and then use it as the initial (*hotstart*) file for the next solution.

7.3.1 Defining Flow and Head

Steady state boundary conditions, as the name implies, do not change with time. For this tutorial, steady state flow and water surface elevation conditions will be defined at nodestrings. First, boundary strings must be created at each end of the stream. The upper boundary string was already created in Lesson 3. To create the lower boundary string (refer to Figure 7-2):

1. Click on the *Create Nodestrings*  tool.

2. Start the string by clicking on the lower left corner node.

3. Hold down the SHIFT key and click on the lower right corner node of the outflow boundary. *SMS* will automatically select all nodes between the two. (The SHIFT key causes the nodestring to progress through the grid by the largest steps possible. Another option would be to hold the CONTROL key which causes the nodestring to proceed counter clockwise around the border of the mesh or the SHIFT and CONTROL keys together which causes the nodestring to proceed clockwise around the border.)

4. Press the ENTER key, or double click on the last node in the nodestring to terminate the string.

After the nodestrings are created, boundary data may be assigned to each.

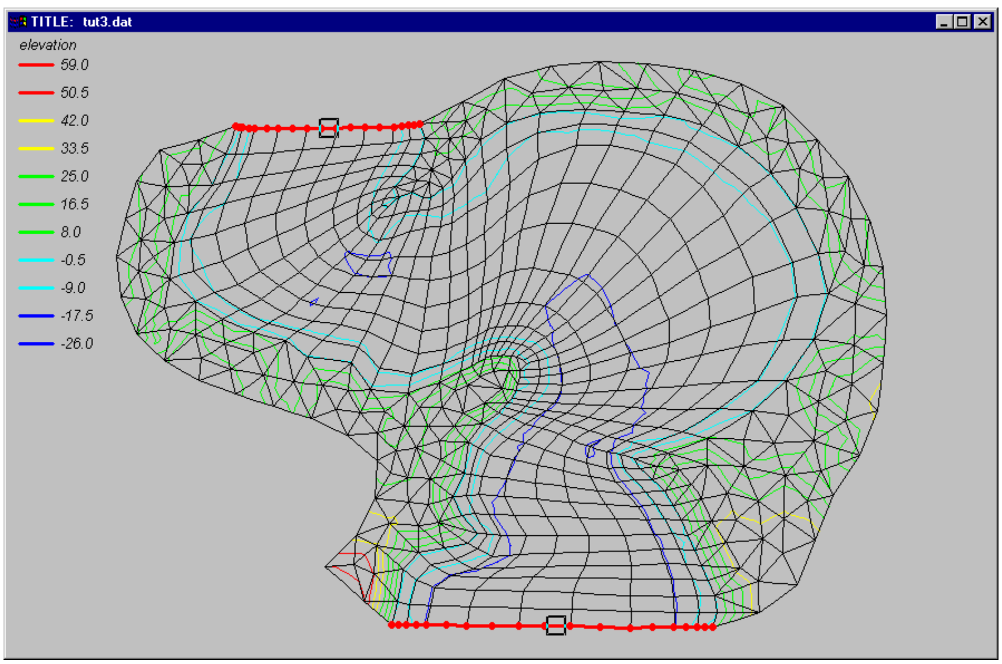


Figure 7-2. Location of the nodestrings.

To assign the steady state flow conditions:

1. Click on the *Select Nodestrings*  tool.

2. Select the upper nodestring by clicking in the box at the center of the string.

3. Select *FESWMS| Assign BC*, click the *Flow* toggle, and make sure the *Normal* option is selected.

4. Assign a value of 9000 (cfs) in the edit box, and click *OK* or press the ENTER key.

5. The selected nodestring is defined as a flow string, and an arrow will appear at the center of the string indicating flow direction.

Now, assign a water surface elevation (head) at the lower nodestring. To do this:

1. Select the lower nodestring.

2. Select *FESWMS| Assign BC* and click the *Water surface elevation* toggle.

3. Assign a value of 60 (feet) in the edit box. Be sure that the option *Essential* is selected.

4. Click the *OK* button or press the ENTER key.

A head symbol will appear at the center of the nodestring, indicating that it is a water surface elevation string.

7.3.2 Creating Weirs, Culverts, Piers and Abutments

With *FESWMS*, flow control structures are easily added to the mesh using the tools to define weirs, piers, culverts, and drop inlets (see the *SMS* Reference Manual). For instructions on how this is done refer to Lesson 8.

7.4 SAVING THE DATA

With *FESWMS* software, data can be saved in a single file, or multiple files may be used. The files used are specified in the *Model Control* dialog. For example, the node and element data may be saved separately from the boundary condition data.

To change options in the *Model Control* dialog:

1. Select *FESWMS| FESWMS Control*.

2. In the *Save File Options* area, select *Write Network File* option. If any of the other three options are *ON*, turn them *OFF*.

3. In the *Output Format* area, choose *FESWMS* version 2.x. Also, make sure the *Solution* option is *ON*, and choose an *ASCII* solution.

4. Click the *OK* button or press the ENTER key.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *FESWMS| Save Simulation*.

2. In the *Prefix for all files* edit box, enter *tut7a*. Then push the *update* button to update all filename prefixes.

3. Check to make sure the full file name path at the top of the dialog is correct and click the *OK* button or press the ENTER key.

The model control options and boundary conditions have been saved to the file "tut7a.dat", while the node and element information has been saved to the file "tut7a.net". If desired, look at the file "tut7a.fil" to see the filenames which were specified. The only input supplied by the user to *FESWMS* is the super file "tut7a.fil", however, *FESWMS* uses all the files created by *SMS*.

7.5 USING AN INiTIAL CONDITIONS FILE

As stated earlier two solutions will be created for this mesh. We have already prepared the *cold start* solution which we will use as an initial conditions (*hotstart*) file for this new solution.

To assign the new steady state flow conditions:

1. Select the lower nodestring.

2. Select *FESWMS| Assign BC*.

3. Assign a value of 57 (feet) in the edit box. Be sure that the option *Essential* is selected

4. Click the *OK* button or press the ENTER key.

A head symbol will appear at the center of the nodestring, indicating that it is a water surface elevation string. This new file will step down the head by a value of 3 feet.

To change options in the *Model Control* dialog:

1. Select *FESWMS| FESWMS Control*.

2. In the *Save File Options* area, select *Read initial condition file* option. Also leave the *Write network file* option on.

3. In the *Output Format* area, choose *FESWMS* version 2.x. Also, make sure the *Solution* option is *ON*, and choose an ASCII solution.

4. Click the *OK* button or press the ENTER key.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *FESWMS| Save Simulation*.

2. In the *Prefix for all files* edit box, enter *tut7b*. Then push the *update* button to update all filename prefixes.

3. Check to make sure the full file name path at the top of the dialog is correct and click the *OK* button or press the ENTER key.

The model control options and boundary conditions have been saved to the file "tut7b.dat", while the node and element information has been saved to the file "tut7b.net". If desired, look at the file "tut7b.fil" to see the filenames which were specified. The only input supplied by the user to *FESWMS* is the super file "tut7b.fil", however, *FESWMS* uses all the files created by *SMS*.

7.6 USING FLO2DH

You are now ready to run an analysis. The analysis module of *FESWMS* is called *FLO2DH*. To run *FLO2DH*, you need the “tut7a.fil” and “tut7b.fil” model files you just saved. To run *FLO2DH*:

1. Path to the directory containing the files mentioned above.

2. Type “*FESWMS tut7a.fil*" to execute the program and specify the problem file.

FLO2DH will open the file "tut7a.fil", and read the data from the files listed. *FLO2DH* will take five to ten minutes to finish the solution. The "tut7a.out" file contains the solution data needed. This solution file can be imported into the data browser in a manner similar to that described in Lesson 10, section 10.3 for reading in *TABS-MD* solution files. Once a solution file has been read, *SMS* processes the data sets created from them in a model independent fashion. Therefore all of the methods for post-processing described in Lesson 10 are applicable to *RMA2*, *SED2D-WES* or *FESWMS*.

We want to use this solution file as an initial file for the next solution.

1. Using a file utility, copy the file “tut7a.out” to “tut7b.ini”. This new file will be used in the next solution and thus should be located in the same directory as those files.

You are now ready to run the next analysis. To run *FLO2DH*, you need the "tut7b.fil" model file and the “tut7b.ini” file you just created. To run *FLO2DH*:

1. Path to the directory containing the files mentioned above.

2. Type “*FLO2DH tut7b.fil*" to execute the program and specify the problem file.

FESWMS will open the file "tut7b.fil", and read the data from the files listed. Analysis will begin from the ending point of the solution in the file “tut7b.ini”.

FLO2DH will take five to ten minutes to finish this solution. The "tut7b.out" file contains the solution data needed.

7.7 CONCLUSION

This concludes the Basic FESWMS Analysis tutorial. We suggest that you now proceed to the 2D Post-Processing tutorial included in Lesson 10 to learn how to view the result of this analysis.

If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 8

Advanced FESWMS Analysis

8.1 INTRODUCTION

This lesson will teach you how to prepare a mesh for a *FESWMS* solution run. You will be using the file "bridge.fil". This file has already been created in a manner consistent with that which has already been outlined in Lesson 2. *FESWMS* supports both 8-noded quadrilateral elements and 9-noded quadrilateral elements. Elements with 9-nodes have been used in this tutorial. The model has also been renumbered. Only the band width renumbering method should be used for *FESWMS* models since the band width method attempts to account for equations used by *FESWMS*. A *FESWMS .fil* file is a type of super file. It contains a list of filenames that may be used by *FESWMS*. The actual input data is stored in the files named in the super file.

Open the file "bridge.fil" which causes *SMS* to read the data shown in Figure 8-1. To do this:

1. Select *FESWMS| Open Simulation*.

2. Select the file "bridge.fil" and click the *OPEN* button or press the ENTER key. If a mesh is open, you will be warned that all existing data will be deleted. If this happens, click the *OK* button or press the ENTER key.

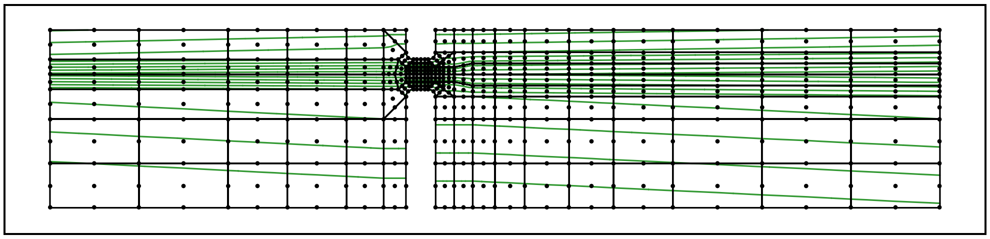


Figure 8-1. The bridge.fil Geometry.

8.2 CREATING MATERIALS

When the mesh was opened, the elements each contained one of four material type ids. The materials were created, but contain default values which must be changed (see the *FESWMS* chapter of the *SMS Reference Manual* for details of what each material parameter represents). To change the material values:

1. Select *FESWMS| Material Properties*.
2. In the upper right corner of the *FESWMS Material Editor* dialog, a graphical image shows what the Manning coefficients at depths are.
3. Select the material named *Material 1*, and be sure that the *ID* is 1.
4. Enter the following values:
    - 0.035 for both *Manning coefficients*
    - 2.0 for *Depth 1*
    - 3.0 for *Depth 2*
    - 0.0 for *Chezy*
    - 20.0 for *Vo*
    - 0.6 for *Cu*
5. Now, select the material labeled *Material 2* and check that it has an *ID* of 2.
6. Enter the same values for *material 2* that were entered for *Material 1*.
7. For Materials 3 and 4, enter the same values with the exception that both *Manning coefficients* are 0.055.
8. Click the *Close* button or press the ENTER key.

You have just assigned values for the four materials in this mesh.

Optional: The materials can be displayed by opening the *Display Options* dialog and clicking the toggle box next to the word *Materials*. See the *SMS Reference Manual* for more about changing the display of materials. Before continuing with this tutorial, turn the *Materials* option *OFF* in the *Display Options* dialog if you had previously turned it *ON*.

8.3 ASSIGNING BOUNDARY CONDITIONS

Boundary conditions define existing parameters, such as flow and water surface elevation. Without proper boundary conditions, a model will not be stable. Further, the values of the boundary conditions, coupled with the geometric attributes, control the solution. After a model is constructed and analyzed, it should be verified. This means that the numerical results must match the physical conditions. To verify a model, we must first be sure that the mesh correctly represents the shape of the region being modeled. We must then be sure that the boundary conditions are accurate measures of the physical conditions. Finally, we can adjust the material parameters to change the performance of the model.

8.3.1 Defining Flow and Head

Steady state boundary conditions, as the name implies, do not change with time. For this tutorial, steady state flow and water surface elevation conditions will be defined at nodestrings. First, two boundary strings must be created at each end of the stream. To do this:

1. Click on the *Create Nodestrings*  tool.

2. Select the upper left node by clicking on it.

3. Select other nodes on the left side of the mesh by holding the SHIFT key and clicking on the lower left node.

4. End the nodestring by pressing the ENTER key of double clicking on the last node.

Repeat the procedure on the right side and two nodestrings will exist (as highlighted in Figure 8-2). After the nodestrings are created, boundary data may be assigned to each.

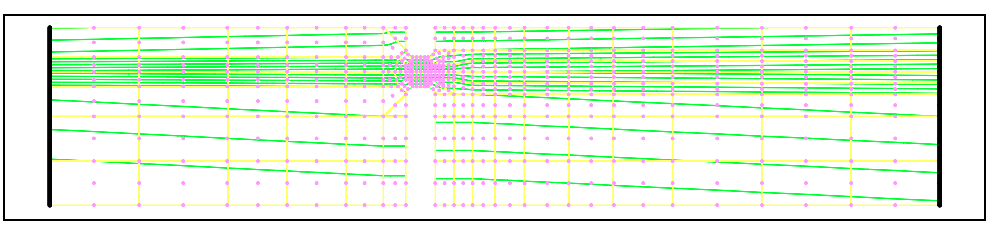


Figure 8-2. Location of the nodestrings in the bridge mesh.

To assign the steady state flow conditions:

1. Click on the *Select Nodestrings*  tool.

2. Select the left nodestring by clicking in the box at the center of the string.

3. Select *FESWMS| Assign BC*, click the *Flow* toggle, and make sure the *Normal* option is selected.

4. Assign a value of 9000 (cfs) in the edit box, and click *OK* or press the ENTER key.

5. The selected nodestring is defined as a flow string, and an arrow will appear at the center of the string indicating flow direction.

Now, assign a water surface elevation (head) at the other nodestring. To do this:

1. Select the right nodestring.

2. Select *FESWMS| Assign BC* and click the *Water surface elevation* toggle. Be sure that the *Essential* option is selected.

3. Assign a value of 812.90 (feet) in the edit box.

4. Click the *OK* button or press the ENTER key.

A head symbol will appear at the center of the nodestring, indicating that it is a water surface elevation string.

8.3.2 Creating Weirs

With *FESWMS*, flow control structures are easily added to the mesh using the tools to define weirs, piers, culverts, and drop inlets (see the *SMS* Reference Manual). For this model, a weir will be defined by five pairs of nodes across the bottom portion of the mesh.

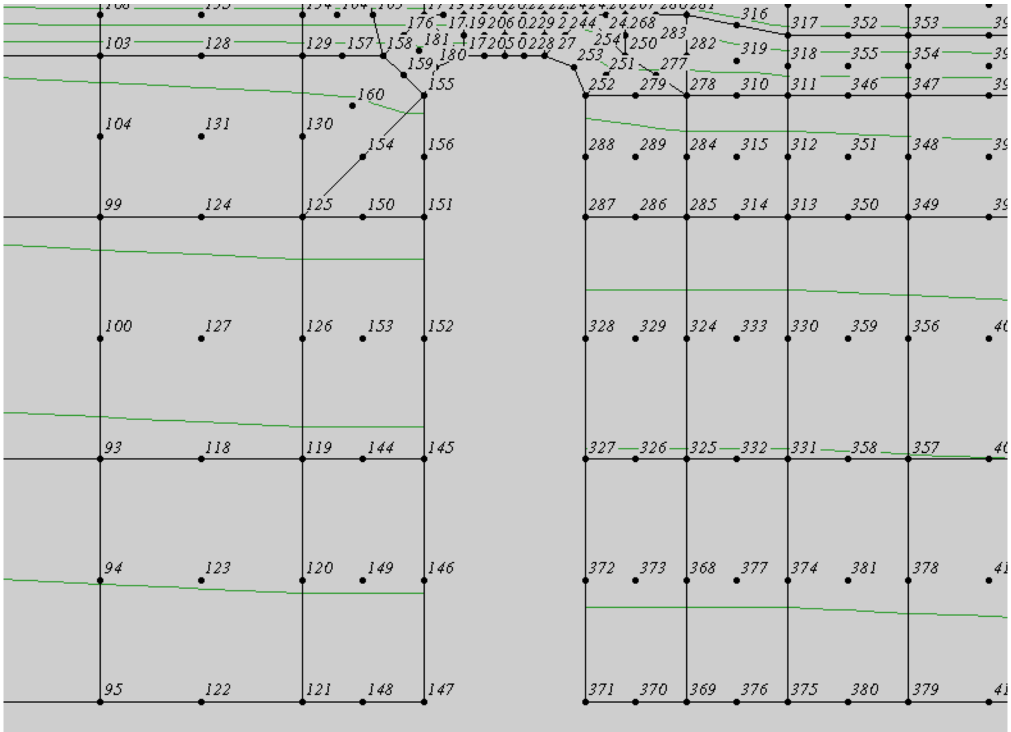


Figure 8-3 Area where weirs will be added.

Turn on the display of node numbers using the *Display Options* dialog. Zoom in on the lower portion of the middle of the bridge (see Figure 8-3). The five node pairs you need to work with are: 151<->287, 152<->328, 145<->327, 146<->372, and 147<->371. In each node couple, the left ID is on the left of the span, while the right ID is on the right side of the span. Select the first set of nodes, numbers 151 and 287 by selecting node 151 and then selecting node 287 while holding down the SHIFT key. Then create a weir between the two. To do this:

1. Select *FESWMS| Weir*.
2. Make sure that node 151 is the *upstream* node, while node 287 is the *downstream* node. If they are opposite, press the switch button.
3. Enter the following values:
   - 0.53 for the *Discharge Coefficient*
   - 37.5 for *Crest Length*
   - 812.5 for the *Crest Elevation*.
4. Click the *OK* button or press the ENTER key.

Select each of the next three pairs of nodes, always making sure that the left node is upstream, and for each pair, enter values of 0.53, 75.0, and 812.5 for the Discharge Coefficient, Crest Length, and Crest Elevation, respectively. Select the last pair of nodes (numbers 147 and 371) and enter the same values that were entered for the first pair.

5. Select the *Frame*  tool from the *Toolbox*.

6. Turn off the display of node numbers using the *Display Options* dialog.

The display should now appear as in Figure 8-4.

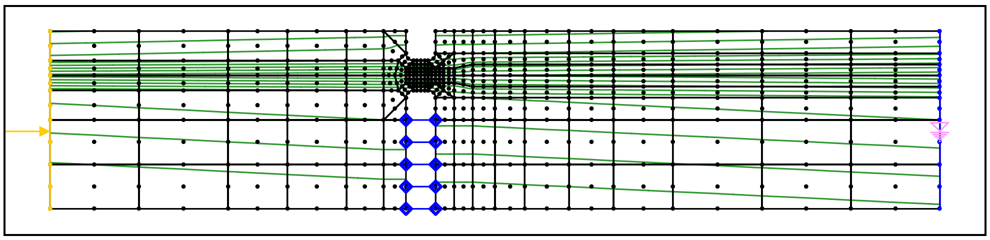


Figure 8-4. The final bridge geometry.

8.4 SAVING THE DATA

With *FESWMS* software, data can be saved in a single file, or multiple files may be used. The files used are specified in the *Model Control* dialog. For example, the node and element data may be saved separately from the boundary condition data. To change options in the *Model Control* dialog:

1. Select *FESWMS*| *FESWMS Control*.

2. In the *Save File Options* area, select *Write Network File* option. If any of the other three options are *ON*, turn them *OFF*.

3. In the *Output Format* area, choose *FESWMS* version 2.x. Also, make sure the *Solution* option is *ON*, and choose an ASCII solution.

4. Click the *OK* button or press the ENTER key.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *FESWMS*| *Save Simulation*.

2. In the *Prefix for all files* edit box, enter *bridge2*. Then push the *update* button to update all filename prefixes.

3. Check to make sure the full file name path at the top of the dialog is correct and click the *OK* button or press the ENTER key.

The model control options and boundary conditions have been saved to the file "bridge2.dat", while the node and element information has been saved to the file "bridge2.net". If desired, look at the file "bridge2.fil" to see the filenames which were specified. The only input supplied by the user to *FESWMS* is the super file "bridge2.fil", however, *FESWMS* uses all the files created by *SMS*.

8.5 USING FLO2DH

You are now ready to run an analysis. The analysis module of *FESWMS* is called *FLO2DH*. To run *FLO2DH*, you need the "bridge2.fil" model file you just saved. To run *FLO2DH*:

1. Path to the directory containing the files mentioned above.

2. Type "*FLO2DH bridge2.fil*" to execute the program and specify the problem file.

FESWMS will open the file "bridge2.fil", and read the data from the files listed. *FLO2DH* will take one or two minutes to finish the solution. The "bridge2.out" file contains the solution data needed. This solution file can be imported into the data browser in a manner similar to that described in the next lesson for reading in *TABS-MD* solution files. Once a solution file has been read, *SMS* processes the data sets created from them in a model independent fashion. Therefore all of the methods for post-processing described in Lesson 10 are applicable to *RMA2*, *SED2D-WES* or *FESWMS*.

8.6 CONCLUSION

This concludes the Advanced FESWMS Analysis tutorial. We suggest that you now proceed to the 2D Post-Processing tutorial included in Lesson 10 to learn how to view the result of this analysis.

If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 9

HIVEL2D Analysis

9.1 INTRODUCTION

This lesson will teach you how to prepare a mesh for analysis and run a solution for HIVEL2D. You will be using the file "bridge_h.2dm", which is a finite element mesh. The geometry has already been created and renumbered following the procedures outlined in Lesson 2.

Open the file "bridge_h.2dm" shown in Figure 9-1. To do this:

1. Select *FILE | Open*.

2. Select the file "bridge_h.2dm". If you still have the geometry open from a previous tutorial, you will be warned that all existing mesh data will be deleted. If this happens, click the *OK* button or press the ENTER key.

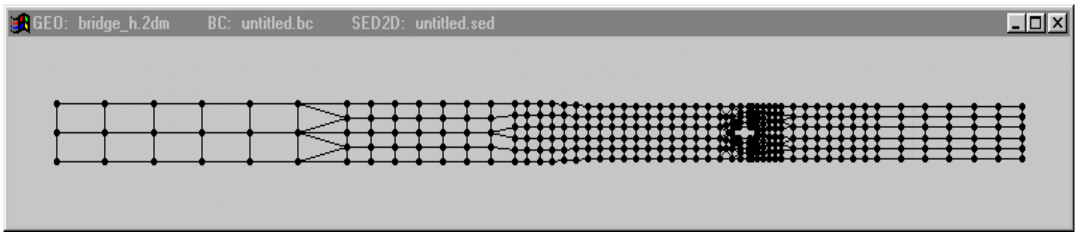


Figure 9-1. The mesh contained in the file bridge_h.2dm.

9.2 CREATING MATERIALS

When this mesh was opened, the elements each contained a material type ID. The materials were created with default parameters which must be changed for this particular mesh. The material properties define how water flows through the element (see the *SMS Reference Manual* for details of what each parameter represents).

There are two materials, however they will both have the same values for this tutorial. To edit the material parameters:

1. Select *HIVEL | Material Properties*.
2. In the *HIVEL Materials Editor*, select the material labeled *Material 1*. Be sure that its ID number is 1 in the *ID* edit box.
3. Enter 0.020 for *n* (Manning's roughness).
4. Now, select the material labeled *Material 2* and check that it has an *ID* of 2.
5. Enter 0.020 for its *n* value also.
6. Click the *Close* button or press the ENTER key.

Each material now has correct parameters associated with it. The materials can be displayed by opening the *Display Options* dialog and turning on the *Materials* toggle. If you desire, turn the materials on to display them.

9.3 DEFINING BOUNDARY CONDITIONS

9.3.1 General Parameters

The geometric mesh is only the first part of the numerical model. We have already defined the material properties associated with the different regions of the mesh. These properties help control how water will flow, and thereby allow us to build a model which matches the physical situation. In addition to the geometry and the material properties, we must define several other boundary conditions. To edit the HIVEL boundary conditions:

1. Select *HIVEL | Model Control*.
2. Enter the necessary values to make all global parameters match those in Figure 9-2. (Select English units first because changing units will automatically update system variables such as gravity).

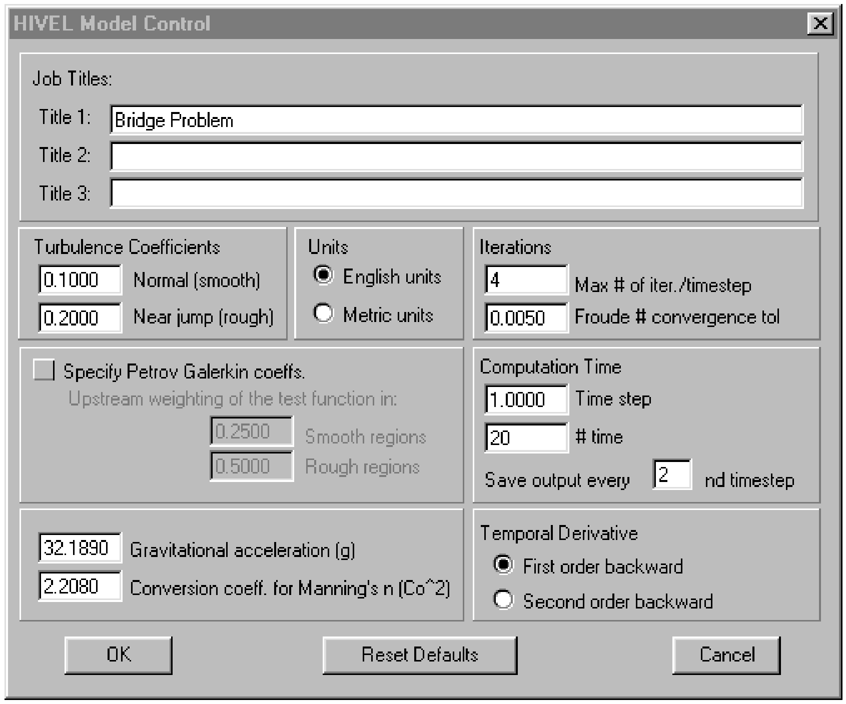


Figure 9-2 HIVEL Model Control dialog

9.3.2 Defining Steady State Flow and Head

For this tutorial, we will define boundary conditions at the nodes of the inflow boundary, as well as along a nodestring at the outflow boundary. The inflow boundary is in supercritical flow, so at these nodes we will specify flow and a water depth (water depth is specified only because the flow regime is supercritical). At the outflow boundary, flow has changed to subcritical (a hydraulic jump has occurred). At this boundary we will specify a tailwater elevation (if flow were still supercritical, we would only specify an outflow boundary location).

To assign the inflow boundary condition:

1. Choose the *Select Nodes*  tool from the *Toolbox* and select all three nodes on the left boundary. You can do this by selecting the nodes one at a time while holding the *SHIFT* key, or by dragging a box around the nodes.

2. Select *HIVEL | Assign BC*.

3. Select the *supercritical* flow option, and specify *Velocity components* of U = 60.0 (fps), V = 0.0 (fps).

4. With supercritical flow, the water depth is also specified. Enter 3.0 (feet) for the depth at these nodes.

5. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

To assign the outflow boundary condition on the right side of the mesh:

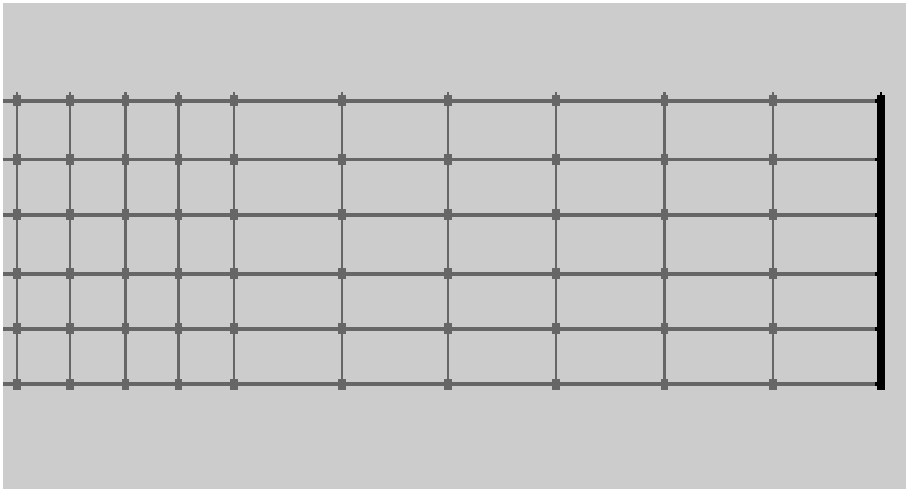


Figure 9-3 Location of the nodestrings in the mesh.

1. Choose the *Create Nodestring*  tool from the *Tool Box* and create a nodestring across the outflow boundary as in Figure 9-3.

2. Choose the *Select Nodestring*  tool from the *Tool Box* and select the nodestring just created.

3. Select the *HIVEL | Assign BC*.

4. Select the *Subcritical* and *Outflow string* options. Enter a value of -30.0 (ft) for the *Tailwater elevation*.

5. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

Generally for *HIVEL2D*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries. See the HIVEL2D Reference manual for more about assigning boundary conditions.

9.3.3 Creating the Hotstart File

Hivel2D must have an initial hotstart file to run a solution. SMS allows you to create this hotstart file either using constant values or a data set that has been previously loaded through the *Data Browser*.

1. Select *HIVEL | Build Hot Start*.

2. Enter 0.00 for the *Time associated with step m*.

3. For both *Step m-1* and *Step m*, enter a *Discharge* of *p* = 50.0 and *q* = 0.0, and a *Water surface elevation* of 3.0.

4. Click the *OK* button or press the ENTER key.

When *HIVEL2D* starts, it will use these values as an initial starting place. It is important to note that at the end of the computations, this file will be over written. Therefore, a backup copy should be created if desired.

9.4 SAVING THE SIMULATION

HIVEL2D uses a geometry file, boundary condition file, and hotstart file written by *SMS* to run an analysis. These files are specified in a *superfile* which is also written by SMS. You must save data that has been created.

1. Select *HIVEL | Save Simulation*.

2. Make sure all available options are turned on. Enter the filename "bridge_h" in the *Prefix for all files* edit box, then click the *Update* button.

3. Enter the filename “wse1” in the *WS solution* edit box.

4. Enter the filename “vel1” in the *Flow solution* edit box.

5. Click the *OK* button or press the ENTER key.

The boundary conditions you specified are now saved. Before continuing with the analysis, check the model for completeness. To do this:

1. Select *HIVEL | Model Check*.

2. Click the *Run Check* button.

The *HIVEL2D* model checker will report one possible warning. The warning is that the mesh has not been renumbered during this session. This is not important because the 2D mesh numbering was valid before being read into this tutorial. Generally, the finite element mesh should be renumbered after being created. Other tutorials discuss creating a finite element mesh.

9.5 USING HIVEL2D

HIVEL2D is the analysis program which computes 2D flow solutions (water surface elevation and velocity) at each node. *HIVEL2D* requires the *superfile* created by *SMS* as input. This superfile tells HIVEL2D where to find the other necessary input files. Note that all of these files must be in the same directory.

If running either on the UNIX, Windows NT, or WIN95 operating systems, the model can be run from inside *SMS*.

1. If running either on the UNIX, Windows NT, or WIN95 operating systems, choose *HIVEL | Run Hivel*.

2. SMS will search for the executable file named “hivel20.exe”. A prompt will appear either letting you know where the executable was found, or saying that it could not be found,

3. Click on the *File Browser* macro to find the *HIVEL* executable.

4. Click the *OK* button or press the ENTER key.

If running on the Windows 3.1 or 3.11 operating systems, the model must be run from a DOS prompt outside of *SMS*.

1. If running on the Windows 3.1 or 3.11 operating systems, open a DOS prompt and path to the directory where your *HIVEL* executable is located.

2. Type the name of the executable (e.g. “hivel20”).

3. When prompted, enter the filename of the superfile you just saved from *SMS*, including the path if located in a different directory (e.g. c:\sms\bridge_h.sup).

HIVEL2D can take between three and ten minutes to run this steady state solution, depending on the speed of your computer. If running from inside *SMS*, the window where *HIVEL2D* ran will go away. The file “wse1.dat” contains the water surface elevation, while the file “vel1.dat” contains the velocity information at each node. These can be imported as *Generic* files through the *Data Browser* (see the SMS reference manual for more information on the *Data Browser*).

9.6 CONCLUSION

This concludes the HIVEL2D Analysis tutorial.

If you wish to exit *SMS* at this point:

- Select the *File | Quit* command.
- Select the *OK* button to confirm.

LESSON 10

2D Post-Processing

10.1 INTRODUCTION

The solutions computed by the *TABS-MD (RMA2, RMA4, HIVEL, SED2D-WES)* and *FESWMS-2DH* software can be viewed in *SMS*. This is called *Post Processing*. In this lesson, you will learn how to import, manipulate, and view solution data. You will need the "ld0.geo" file and the solution "ld0.sol" created in Lesson 5. If you did not perform the tutorial presented in Lesson 5, use the "ld0.sol" file that shipped with *SMS*.

10.2 DATA SETS

After an analysis is run, solution data is printed to a file in the form of *data sets*. Each data set contains a certain type of solution data at each node of the mesh. *RMA2* and *FESWMS* create data sets representing quantities such as flow velocity and water depth. *RMA4* creates data sets representing constituent concentrations. *SED2D-WES* creates data sets representing sediment concentration and change in bed elevation. When *SMS* imports a solution file, it creates a data set for each quantity in the file. For steady state solutions, a single data set is created for each quantity. For dynamic solutions, a data set is created for each time step. Some data sets consist of scalar data, while others consist of vector data.

An *RMA2* solution contains three scalar data sets and one vector data set. The scalar sets are water depth, water surface elevation, and water velocity magnitude. The vector set is water velocity.

An *RMA4* solution contains one data set for each substance constituent that was specified.

A *SED2D-WES* solution contains four scalar data sets. These are water depth, bed shear stress, bed elevation change, and sediment concentration.

A *FESWMS* solution contains two scalar data sets and one vector data set. The scalar data sets are water depth and water velocity magnitude. The vector data set is water velocity.

Steady state scalar data can be viewed as contours. Steady state vector data can be viewed as vectors. Other ways of viewing the data sets are *Film Loops* and *Gages*, which are described later in this lesson.

10.3 USING THE DATA BROWSER

All solution files are imported into *SMS* through the *Data Browser* dialog. Solutions may be imported only if the mesh corresponding to the solution is already loaded. If the mesh is edited so that it no longer corresponds to a solution file, *SMS* will detect this and not allow the solution file to be imported. In addition, as soon as the mesh is edited, the existing data sets loaded in *SMS* are deleted.

To import the "ld0.sol" *RMA2* solution file:

1. If the "ld1.geo" mesh is not opened, open it as described in Lesson 5.

2. Select *Data | Data Browser* or click on the *Data Browser*  macro.

3. Click on the *Import* button. From the *Import Data Set* dialog select the *TABS file* button and click *OK* or press the ENTER key. *Note: the TABS option is used to import RMA2, RMA4, and SED2D-WES solutions. The FESWMS option is used to import FESWMS solutions*.

4. From the *Open File* dialog find and highlight the file "ld0.sol", and click the *OPEN* button or press the ENTER key.

5. The file "ld0.sol" was the solution to the "ld1.geo" mesh with dynamic boundary conditions, and it is therefore a dynamic solution file. When a dynamic solution file has been imported, the current data set’s time steps are displayed in the *time step* text window.

6. Select “velocity magnitude” in the *Scalar Data* text window and “velocity” in the *Vector Data* text window.

7. Select a desired time step for both the scalar data set and the vector data set.

8. Click the *Done* button or press the ENTER key to exit the *Data Browser*.

After exiting the *Data Browser* dialog, the display is updated. If contours or vector arrows are being displayed, the new display will correspond to the data in the newly selected time step/data sets. Note that the displayed contours and vectors can be changed in color, density, etc. See the *SMS* Reference Manual for information on changing the display of contours and vectors.

10.4 CREATING NEW DATA SETS WITH THE DATA CALCULATOR

SMS has a powerful tool for creating new data sets from existing data sets, called the *Data Calculator*. Using the *Data Calculator*, you can manipulate solution data to get other desired solutions. You will now create a data set which contains the Froude Number at each node. The Froude number is given by equation:

$$\text{Froude Number} = \frac{\text{Velocity Magnitude}}{\sqrt{\text{gravity} * \text{water depth}}}$$

With the *Data Calculator*, you can create a data set of the Froude Number by performing each necessary operation one by one. Choose *Data* | *Data Calculator* to open the *Data Calculator* dialog. *SMS* allows for complete input of the equation on the equation line. This can be done as follows:

1. Under the *Time* options click on the *Use all time steps* toggle. This will compute the function for every time step value.

2. Choose the *velocity mag* data set then Click the *Add to Expression* button. Notice that expression now contains the letter “c” corresponding to the *velocity mag* data set.

3. Select the *divide* “ / “operation from the operations palette.

4. Select the *square root* operation from the operations palette.

5. Using the mouse click in the expression edit box and delete the closing parenthesis “) “.

6. Choose the *water depth* data set and Click the *Add to Expression* button. Notice that expression now contains the letter “*b*” corresponding to the *water depth* data set.

7. Select the *multiply* “ * “operation from the operations palette and enter a value for gravity. This mesh uses English units, so enter a value of 32.17405.

8. Select the *closing parenthesis* “) “operation from the operations palette.

9. In the *Results* edit box enter the name “froude”, then click the *Compute* button.

This may take a few minutes depending upon the speed of your computer.

10. Select *Done* or press the ENTER key to exit the *Data Calculator*.

The Froude Number data set now contains the Froude number as calculated with the Data Calculator. It may be treated just as any other dynamic scalar data set.

10.5 CREATING FILM LOOPS

A film loop is an animation in *SMS* which displays selected time steps of the selected data set in sequence and represents the solution for the entire mesh. Flow trace film loops use velocity vector data sets to trace the path that particles of water follow through the flow system. Only the zoomed in portion of the mesh will be included in the film loop. For this tutorial, you should zoom into the portion of the mesh shown in Figure 10-1. To do this:

1. Select the *Zoom*  tool from the *Toolbox*.

2. Click and drag the mouse to create a box about the area to zoom.

3. Let go of the mouse button and the display will update.

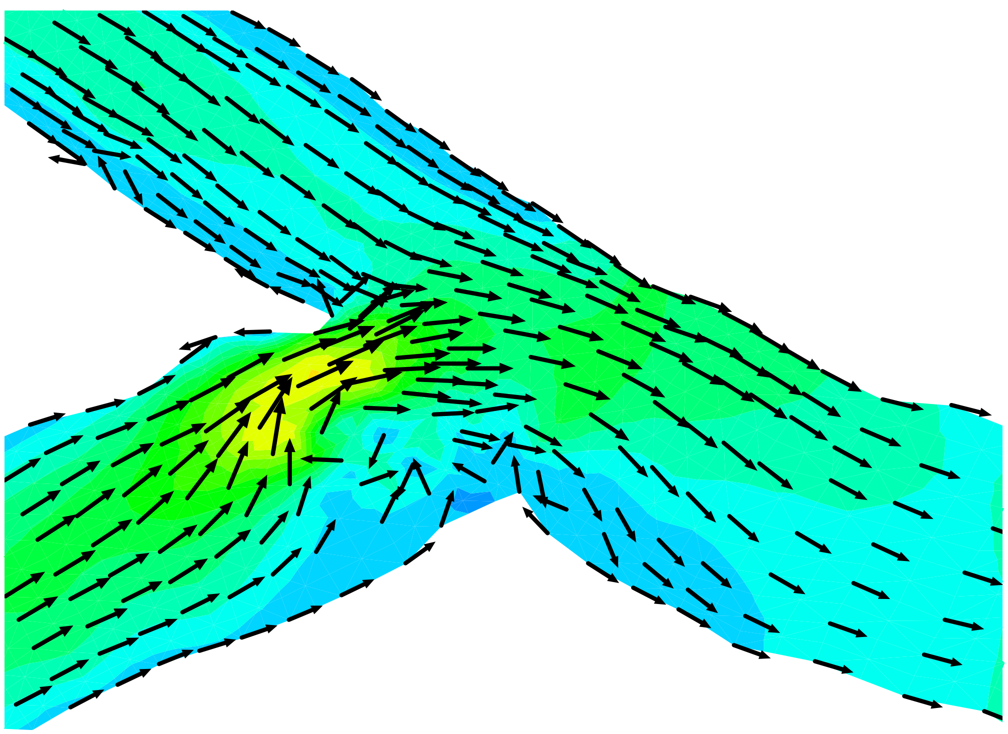


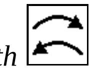
Figure 10-1. The area about which to zoom for the film loops.

10.5.1 Creating and Running a Vector Set Film Loop

The first film loop you will create will show the flow direction in the mesh. To create and run the film loop:

1. Select *Data | Film Loop* and click the *Setup* button.
2. Click on the *Data Browser* macro.
3. Select “velocity magnitude” in the *Scalar Data Sets* text window and “velocity” in the *Vector Data Sets* text window.
4. Select a desired time step for both the scalar data set and the vector data set.
5. Click the *Done* button or press the ENTER key to exit the *Data Browser*.
6. Click the *Display Options* macro button and turn everything off except *Velocity vectors* and the *Mesh boundary*.
7. Click the *Options* button next to *Velocity vectors*.
8. In the bottom right corner of the *Vector Options* dialog, turn the *Display corner node vectors only* option on.
9. In the upper right area of the *Vector Options* dialog, select the *Setup Colors* button. This will open the *Color Ramp Options* dialog.
10. In the *Color Ramp Options* dialog select the *Use range of hues* option and select *OK* or press the ENTER key.
11. Click the *OK* button or press the ENTER key to exit the *Vector Options* dialog, then exit the *Display Options* dialog, still leaving the *Film Loop Setup* dialog up.
12. Change the *Size (% screen)* scroll bar to 75 %.
13. In the *Data Options* area of the dialog, select the *Vector data set* toggle and select all time steps in the *time step range* display. (The simulation should run from time 0 to time 570). (The default is normally to use all time steps.)
14. Choose the *Use constant interval* option and make sure a value of 10 is used.
15. Click the *OK* button or press the ENTER key to exit this dialog, and create the film loop.

SMS will create the film loop, and a prompt in the *Help* window shows the task being performed. When the film loop has been generated, click the play button.

The film loop frames previously generated will animate from one frame to the next. During the animation, speed and play mode can be changed. Change the animation speed (dependent on the speed of the computer) using the speed scroll bar. Change the play mode by clicking the *loop*  icon or the *back-forth*  icon. To stop the animation process, click the stop  button. The step  button can be used to manually step one frame at a time. A single frame specified by the frame scroll bar or edit field can be viewed. When you are finished viewing the film loop, you may save it if you wish by clicking the Save button and entering a file name. After saving a film loop, it may be read in and viewed at some future time. See the *SMS Reference Manual* for more information about viewing film loops.

10.5.2 Creating and Running a Scalar Set Film Loop

With the *Film Loop* dialog still up, you can create a new film loop. This time, the film loop will show water velocity at each time step. To create this film loop:

1. Click the *Setup* button. If you have not saved the previous film loop, you will be warned. If this happens and you want to save the film loop, click *No*, then save it as explained above. Otherwise, click *Yes*.

2. Click the *Display Options* macro button and turn everything off except *contours*.

3. Click the *Options* button next to *contours* to open the *Contour Options* dialog.

4. In the *Contour Method* area, select the *Color fill between contours* toggle.

5. Click the *Color Opts* button and turn the *Show color legend* option *OFF*.

6. Exit the *Contour Color Options*, *Contour Options*, and *Display Options* dialogs by clicking the *OK* button or pressing the ENTER key in each.

7. Make sure the *Scalar data set* toggle is selected and the *Vector data set* toggle is not.

8. Change the *zooming* to 50 % of the screen.

9. Click *OK* or hit the ENTER key to allow *SMS* to generate this film loop.

After *SMS* has created the film loop, you may view it by using the same procedures described in the previous film loop. When you are done viewing it, you may save it as described above. However, do not close the *Film Loop* dialog -- you will still use it for the next film loop.

10.5.3 Creating and Running a Flow Trace Film Loop

A flow trace film loop can be created if a vector data set, either dynamic or steady state, exists in memory. Flow traces introduce massless particles and follow their paths through a vector field. If the vector data set is dynamic, the user decides whether all time steps are used for the film loop or only the current time step. With steady state vector data sets, only one time step exists, so the film loop will represent this steady state case. Note that a flow trace will take upwards of 5 to 15 minutes to generate, depending on the speed of your machine. To create and run a flow trace film loop:

1. With the *Film Loop* dialog still up, click on the *Setup* button.

2. On the left side of the *Film Loop Options* dialog, turn the *Flow Trace* toggle *ON*.

3. Select the *Use specified time steps* toggle to trace all time steps of the current vector data set, which is water flow.

4. This time, use 25% for the screen size.

5. All other options should be kept at their default values for this tutorial. See the *SMS Reference Manual* for more details about flow trace options.

6. Click the *OK* button or press the ENTER key.

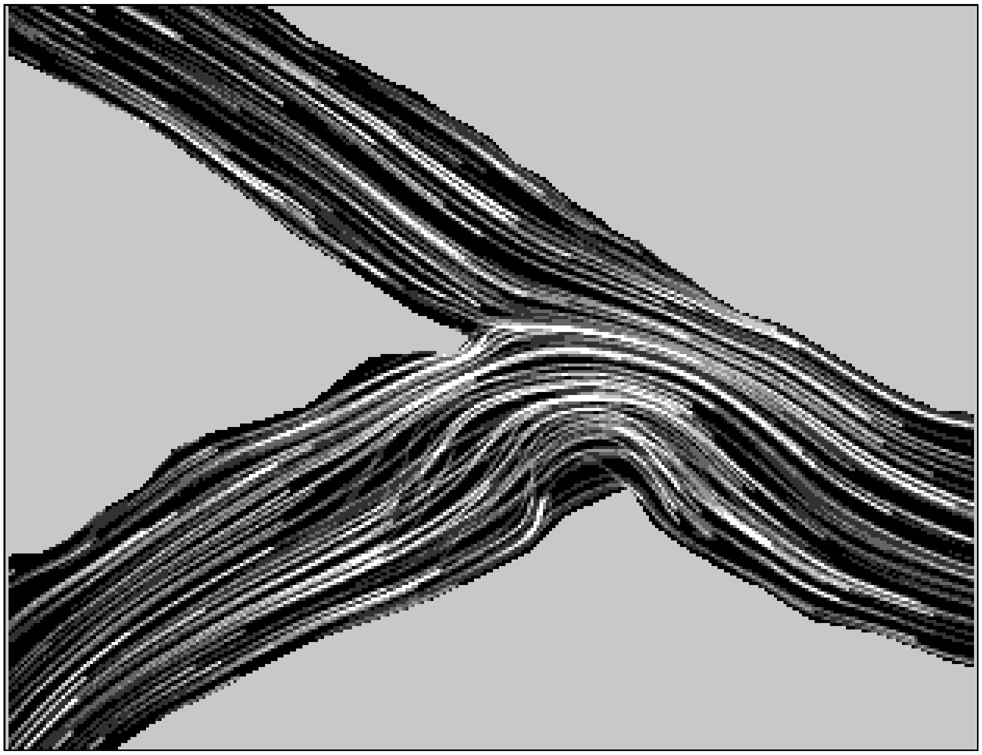


Figure 10-2. One frame from the ld1 flow trace.

As with the other film loops, a prompt shows the task being performed. After *SMS* has created the flow trace, view it as desired. Figure 10-2 shows a frame from this flow trace. When you are finished viewing the flow trace, save it if you wish, and click the *Done* button or press the ENTER key.

Before continuing with the tutorial, you should frame the image on the screen and make sure nodes and elements are displayed. To do this:

1. Select *Display* | *Frame Image* or click the *Frame*  macro.

2. Bring up the *Display Options* dialog and make sure the *Node* and *Element* toggles are selected.

10.6 CREATING GAGE PLOTS

Dynamic solution data can also be displayed in graph form using a *gage*. A gage is an *SMS* tool which is placed at a specific location in the mesh. A *gage plot* allows the user to select specific data sets at selected gage locations and plot the variation of those data sets verses time. A dynamic solution must be resident in the *Data Browser*. To create a gage plot:

1. Select the *Create Gage*  tool from the *Toolbox*. Create a few gages by clicking at various locations in the mesh.

2. Select the *Select Gage*  tool from the *Toolbox*. Select one of the gages you created by clicking on the gage or dragging a box around it.

3. Select *Display | Show Plot Window* to open the *Plot Window*. After about a minute, a plot of all current dynamic data sets will be displayed in the *Plot Window*.

4. Select *Data | Plot Manager*. The *Plot Manager* dialog will open.

5. Up to five plots can be visible at once. Make the second plot visible by selecting the Untitled #2 plot and clicking the *Make Visible* button. The *Plot Window* will be updated showing two plots of all current dynamic data sets.

6. Select the Untitled #1 plot and click the *Curves* button. The *Gage Plot Curves* dialog should then appear, and you can then choose which curves will be shown in the selected plot.

7. Click the *<-All* button to move all *Plotted Curves* to *Available Curves*. This turns all of the curves *OFF*. Now select the ‘water depth’ data set in the *Available Curves* text window and click the *Selected->* button. This turns the water depth curve *ON*. Now, only water depth values will be plotted on the first graph.

8. Click the *OK* button or press the ENTER key to close this dialog.

9. In the *Plot Manager* click the *Plot Options* button. Type “Water Depth” in the *Title 1* field of the *Plot Options* dialog then click the *OK* button or press the ENTER key.

10. Select the second visible plot and click the *Curves* button.

11. Make only the ‘velocity’ data set visible, then exit the *Gage Plot Curves* dialog.

12. Click the *Plot Options* button and type “Velocity” for the title of this plot. Then, exit the *Plot Options* dialog.

13. Click the *Close* button or press the ENTER key to exit the *Plot Manager*.

The gage plots shown represent the selected dynamic data set data at the selected gage’s location. If you deselect a gage, the plots will go away. You can select more than one gage and the plots will be updated with the previously selected data sets at each gage. Hold down the SHIFT key and click on more than one gage to select multiple gages. A gage can also be moved anywhere in the mesh. To move a gage:

1. Make sure the *Select Gage*  tool is selected.

2. Click on a gage and drag it to a new location.

3. To change a gage's direction, click and drag its arrowhead to a new direction.

After moving a gage or changing its direction, the gage plot(s) associated with it will be updated. To close the *Plot Window*, select *Display* | *Hide Plot Window*.

10.7 CONCLUSION

This concludes the 2D Post-Processing tutorial. If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.

LESSON 11

WSPRO Analysis

11.1 INTRODUCTION

WSPRO is a water-surface profile computation model that can be used to analyze one-dimensional, gradually-varied, steady flow in open channels. *WSPRO* also can be used to analyze flow through bridges and culverts, embankment overflow, and multiple-opening stream crossings.

In this lesson a one-dimensional model will be created and run through the *WSPRO* analysis package. A bridge section will then be added to the model and *WSPRO* analysis will be repeated. The uncontracted and contracted profiles will then be compared. The bridge will then be modified to design for allowable backwater and be reevaluated.

11.2 Starting the River Module

The river module utilizes information from both the river window and the plot window. Therefore all these windows should be in view before creating the model.

1. Click on the *River Module*  icon.
2. Select *Display | Show River Window*.
3. Select *Display | Show Plot Window*.

Because the graphics window will not be used in one-dimensional model creation it may be advantageous to enlarge the plot window and shrink the graphics window.

11.3 Cross Sectional Data

11.3.1 Opening a Cross Section File

We will start with two cross sections that have been previously defined. These sections represent the approach and exit sections for a river reach in which a bridge is to be constructed. To read in the sections:

1. Select *WSPRO | Open Simulation*.

2. Select the file "basic.dat". If you have river data defined in *SMS* you will be warned that all existing river data will be deleted. If this happens, click the *OK* button or press the ENTER key.

The cross sections (simulation) are now loaded and the river window shows the icons for the different cross sections (see Figure 11-1).

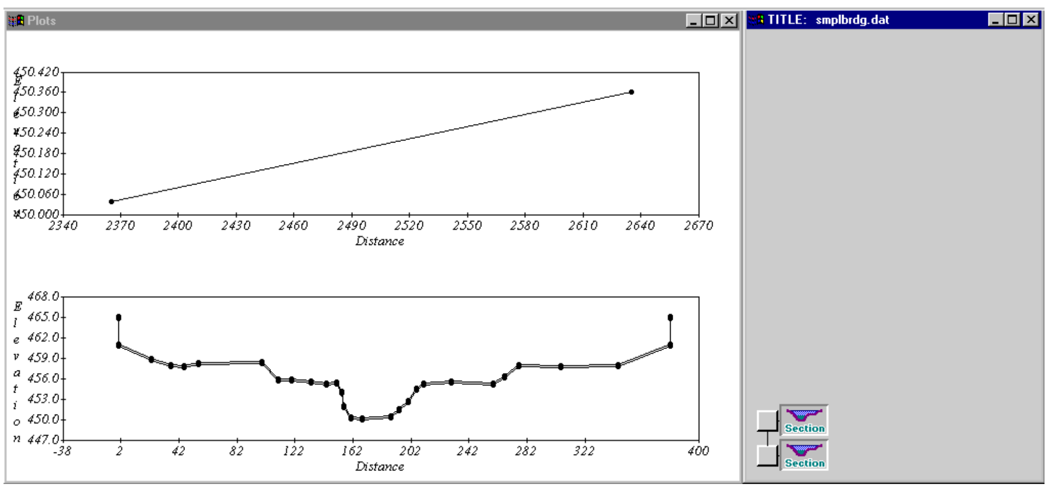


Figure 11-1 Plot window and River window

Clicking on a section causes it to be selected. All selected sections are displayed in the plot window. Double clicking on a section invokes the *Section Editor* dialog. Multiple sections can be selected by dragging a box around the desired sections, or by holding the SHIFT key down and selecting the desired sections. (if the SHIFT key is held down while clicking on a selected section, it is unselected). Note: when a file is read in, all sections are selected by default.

11.3.2 Inserting an Interpolated Cross Section

We need to create a new station where the bridge will be located. This new station will be between the two existing stations. *SMS* will interpolate between the approach and exit sections to create the full valley section under the bridge. To do this:

1. Select the *Create Station*  tool from the *Toolbox*.

2. Click in between the two existing stations in the *River Window*. This will add a station with a new section. SMS will prompt you for the *Section Reference Distance* (SRD) of the new cross section.

3. Assign a location value of 2485 feet for the new SRD. (this location will be explained later).

The new cross section now exists. Its geometry is an interpolation of the existing cross sections. *SMS* automatically invokes the *Section Editor* dialog to define properties, attributes and geometry for the new section.

For this case, the geometry is okay as interpolated. We do, however, need to create section break points to specify where the roughness of the bed changes. We want these break points to match those used in the approach and exit sections.

Section Break Points

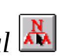
1. Select the *Create a Section Break*  tool from the *Section Editor*.

2. Add section breaks by clicking in the plot window as close to the desired point as possible. Breaks should be added at x locations of 99, 150, 210 and 276 feet.

3. Select the *Select a Section Break*  tool, with this tool you can move a section break location by dragging it in the plot window, or by entering the exact x value in the edit window.

Next we need to specify the material type for each sub-region (between SA points) of the section.

Material Properties

1. Select the *Select section to assign material*  tool from the *Section Editor*.

2. View currently assigned materials by clicking in the plot window between section breaks. The material of that section will be displayed on the *material type* button in the lower right corner of the dialog. These all default to the first material defined for the simulation, (ID=1). For this lesson, material 1 is named “n_0.055_0.050”.

3. Select the material type button to bring up the materials editor. This allows you to assign a particular material to the selected section.

4. Use this tool to assign materials to the newly created sections. Material ID's of the new sections are left to right 1-2-3-4-1.

Now we have defined the necessary parameters for the full valley section. Close the *Section Editor* by clicking the *Done* button or pressing the ENTER key.

11.4 Defining Global Parameters

Global model parameters and boundary conditions, such as flow and water surface elevation, must be supplied for *WSPRO* to compute a profile.

1. Select *WSPRO | WSPRO Run Control.*
2. In the *Flow Sets* section of the dialog, select the EXIT section and click the  button to add it to the flow sets. This allows us to specify flow rates at the EXIT section.
3. In the *Profile Control Data* section of the dialog click the *Add* button. This defines a profile for *WSPRO* to compute. *WSPRO* can compute multiple profiles in a single run, but for this lesson, we will just look at one. For each profile, the user must supply the flow rate (Q) at each section, a computation direction, and either a friction slope or initial water surface elevation.
4. Assign a flow(Q) of 5500 cfs for the exit section and a initial friction slope of 0.002 ft/ft. Also make sure the *Upstream* option is selected, indicating subcritical flow.
5. Exit the dialog by clicking on the *OK* button or pressing the ENTER key.

11.5 Model Check

The boundary conditions are now specified. Before continuing with the analysis, check the model for completeness.

1. Select *WSPRO* | *Model Check*.
2. Click the *Run Check* button.

The *WSPRO* model checker will report one possible error. The warning is that the GR point in the XS section is to close to vertical. This is due to the vertical sides of the cross-sections. *WSPRO* will automatically adjust these values during profile computation. Vertical points are moved 0.1 units apart so that there are no vertical edges.

11.6 Saving WSPRO Simulation

After the river model has been defined it is ready for *WSPRO* analysis. Before analysis we must save the simulation file.

1. Select the *WSPRO | Save Simulation* option.
2. Save the simulation as “basicout.dat”.
3. Click the *OK* button or press the ENTER key to save the file.

11.7 Run WSPRO Analysis

WSPRO uses the current simulation file to calculate the specified water surface profiles and tables. The simulation file contains the sectional properties along with material properties and global parameters. To run *WSPRO* on the current simulation:

1. Select *WSPRO | Run WSPRO.*
2. If the data in the simulation has changed *SMS* will prompt the user to save the data before running *WSPRO*. Click on the *OK* button to save and run or press the CANCEL button to interrupt execution.
3. *SMS* looks through the execute path for the *WSPRO* executable and displays the current location. If it is not found, or if *SMS* finds a version that you don’t want to use, click on the File icon to find and select the proper executable.
4. Click the *OK* button to run *WSPRO*.

11.8 View Uncontracted Profile

If you have closed your simulation you must first open it up as described in section 11.2 of this tutorial. *WSPRO* creates two output files. The “.plt” file is read and interpreted by *SMS*. The “.lst” file is a text file that stores tables of solution variables and can be viewed through a text editor.

11.8.1 Viewing WSPRO results from the .lst file

1. Select *WSPRO | View Data File.*
2. Select the basicout.lst file from the file browser.
3. Select the text editor to view the file. The default editor is notepad for PC platforms and VI for UNIX platforms. Click the *OK* button.

The ".lst" file contains an echo of the input data, the standard profile solution data, and other user specified tables. The top portion of the file echoes the input parameters and includes warnings and error messages generated by *WSPRO*. At the end of the file, solution data is output for each of the three cross sections. Each section is labeled to show variables such as WSEL (water surface elevation). User defined tables are also included at the end of the file.

11.8.2 Viewing results graphically in SMS

1. Select all three cross sections in the river window so that a profile can be shown.

2. Select *WSPRO | Display Options*. This opens the *WSPRO Display Options* dialog.

3. Click the *Import* button.

4. Highlight and select the file "basicout.plt". Click on the *Open* button or press the ENTER key.

With the solution file imported, plots of each of the variables listed on the left side of the dialog may be created. In this case no constriction such as a bridge or culvert exists in the model. Therefore the contracted variables have no meaning for this profile.

By default, three variables are turned on in addition to the Thalweg. These include the water surface elevation (WSEL) shown in blue, the critical water surface elevation (CRWS) shown in red, and the energy grade line (EGL) shown in green.

The colors and symbols used for each variable plot may be modified by clicking on the symbol to the left of the plot toggle box.

Exit the dialog by clicking the *OK* button or pressing the ENTER key. Upon exiting, the plot window will show plots for all functions that are set to be visible.

The plot can be manipulated by opening the *Section Editor* dialog and using the *Zoom image*  tool and the *Pan*  tool.

11.9 Addition of a Bridge

11.9.1 Design loop to meet design criterion.

We now want to add a bridge at the cross section that was created in section 11.3.2 of this lesson.

1. Select the *Create Bridge* tool and click on the middle cross section. This creates a default bridge.

   The default bridge is assigned an elevation equal to the average elevation of the cross-section. Its maximum length is one fourth as long as the cross-section is wide.

   Once again, *SMS* automatically invokes the *Section Editor* to allow the user to change the default parameters.

2. In the *Section Editor*, click the *Attributes* button. This opens the *Bridge Section* dialog which allows the design of a bridge structure.

3. In the *Bridge Section* dialog change the abutment and embankments to vertical and specify a deck width of 30 feet. The SRD for the bridge is at the downstream face of the bridge. With a bridge 30 feet wide, it is centered between its approach and exit with an SRD of 2485.

4. Select the *Parameters* button. This brings up the *Bridge Component Model Parameters* dialog.

5. In the *Abutment* section of the dialog, make sure the bridge is centered and assign a *length* of 120 feet. Constrain the abutment locations to 135 feet (XCONLT) and 225 feet (XCONRT). This keeps the bridge abutments out of the main channel.

6. Specify a *deck elevation* of 462 feet.

The length of the bridge and deck elevation are two variables that can be altered as we design the bridge to meet specifications. We will be modifying this bridge length later in this tutorial.

After these values are entered:

1. The model should be run through the model checker (as in section 11.5).

2. Then the simulation should be saved as wp_br1.dat (as in section 11.6).

3. Finally it should be run through *WSPRO* as outlined in section 11.7 of this lesson.

11.9.2 Check Backwater and Deck clearance

In the last section, we created a bridge. We now want to design the bridge to generate one foot of backwater at the specified flow. In practice, this may take several design modifications before a final design is reached. The design can be visually evaluated by examining the plots of the data variables. Exact values can be obtained in the .lst file.

Visual Analysis

1. Import the new .plt file as indicated in section 11.8.2 of this lesson.

   With the addition of the bridge, results are now calculated for uncontracted and contracted profiles. The contracted plots have hollow boxes at the data points while the uncontracted have filled boxes.

2. Open the *WSPRO | Display Options* and turn off all variables except the two WSEL plots. This requires turning off the EGL and CRWS functions.

3. Turn off the cross section viewing in the lower right portion of the dialog.

In the plot window two profiles are plotted showing the contracted and uncontracted WSEL. The critical points are the differences in these lines at the approach (backwater) and the WSEL at the bridge. At the bridge section you will see two dots on the contracted water surface elevation, these dots represent the upstream and downstream faces of the bridge.

Choose *WSPRO | Edit Section* and zoom in around the approach section WSEL points (far right points of plot). By placing the cursor over the pints, the water surface elevation can be seen in the *View tools* section of the dialog. The contracted WSEL is approximately 461.1 ft. and the uncontracted is approximately 460.5 ft. This indicates that the bridge generates about 0.6 ft of backwater.

Using the .lst file

We can look at the values in the .lst file to get exact number for the profile plot points.

1. Open the new .lst file as shown in section 11.8.1.

2. Scroll or move down to the end of the file.

3. Find the string “Beginning 1 Profile Calculation(s)”. This is followed by the profile data for the simulation. The first data block for the APPR section shows that the uncontracted WSEL is 460.513 ft. The second block for the APPR section has data for the contracted profile. It shows the WSEL as 461.104 ft. This confirms the visual results.

11.10 Changing Model Parameters

FEMA allows bridges to generate 1.0 ft of backwater. Therefore we can shorten the bridge and still have a valid design.

1. Open the *WSPRO | Edit Section* dialog and click on the arrows until the bridge is the current section (blue box around bridge section in *River Window*).

2. Select the *Attributes* button to edit the bridge.

3. Select the *Parameters* button to get to the bridge length.

4. Change length to 100.0 ft.

5. Exit all the dialogs using *OK* and *Done* buttons.

Resave the file as “wp_br2.dat” and run *WSPRO* again. By importing the new “wp_br2.plt” file, we can visually observe that the WSEL for the contracted profile at the approach is now approximately 461.4 ft. The .lst file shows 461.422 ft. This indicates a backwater of 461.422 - 460.513 ≅ 0.9 ft. This indicates the bridge could be narrowed again if more precision is required.

11.11 Conclusion

The bridge length is only one parameter on the bridge design. The bridge deck elevation, channel roughness, abutment types, etc. all can be modified to affect the results of the analysis. Other sections such as culverts and roadways can be defined to model other aspects of the cross section. Culverts are used to add conveyance, while roadways are used to model overtopping during flood conditions.

These and several other options for *WSPRO* and the *SMS* interface to *WSPRO* are included in the *SMS Reference Manual* and the *WSPRO Users Manual*.

This concludes the WSPRO Analysis tutorial. If you wish to exit *SMS* at this point:

- Select the *File* | *Quit* command.
- Select the *OK* button to confirm.